

Table of Contents

TRIFLEX® Windows User Manual

| | |
|--|-----------|
| Introduction to TRIFLEX® Windows | Chapter 1 |
| Tutorial | Chapter 2 |
| 2 TUTORIAL | 5 |
| 2.1 Getting Started | 5 |
| 2.1.1 Main Screen Layout | 6 |
| Status Bar | 7 |
| 2.1.2 Commands for Graphical Operations | 8 |
| 2.1.3 Accessing Data from Piping Model | 9 |
| 2.1.4 Using the Manual and Help Command | 10 |
| 2.2 Opening and Importing Example Piping Model Files | 10 |
| 2.2.1 Processing a Previously Built Piping Model | 12 |
| 2.2.2 Printing Output Reports | 13 |
| 2.2.3 Append, Insert and Replace Mode | 14 |
| 2.3 Coding the New Sample Problem | 16 |
| 2.3.1 Define the Problem | 16 |
| 2.3.2 Starting Triflex Windows | 18 |
| 2.3.3 Coding the Components | 21 |
| 2.3.3.1 Anchor Data Point 1000 | 21 |
| 2.3.3.2 Pipe Data Point 1010 | 26 |
| 2.3.3.3 Branch Connection Data Point 1020 | 28 |
| 2.3.3.4 Pipe & Anchor Data Point 1030 | 30 |
| 2.3.3.5 Elbow Data Point 1040 | 32 |
| 2.3.6 Branch Connection | 33 |
| 2.3.3.6 Branch Connection Data Point 1050 | 33 |
| 2.3.3.7 Valve Data Point 1060 | 36 |
| 2.3.3.8 Pipe & Anchor Data Points 1060 through 1070 | 38 |

| | |
|---|----|
| 2.3.4 Executing a Static Analysis | 42 |
| 2.3.5 View Run Output | 43 |
| APPENDIX A | 52 |
| APPENDIX B..... | 54 |
| APPENDIX B..... | 58 |

TRIFLEX® Windows Theory Manual

| | |
|---|-----------|
| Outputs | Chapter 6 |
| Rotating Equipment Compliance Reports | Chapter 7 |
| Triflex® Windows Piping Code Compliance Reports | Chapter 8 |
| Triflex® Windows Dynamic Capabilities | Chapter 9 |
| Related Engineering Data | Appendix |

List of Figures

| | |
|--|----|
| Figure 2.1.0-1 Demo IU1.dta Example | 6 |
| Figure 2.1.1-2 Status Bar Indicator | 7 |
| Figure 2.1.2-1 Graphic Toolbar Buttons | 8 |
| Figure 2.1.3-1 Viewing Anchor Component Properties | 9 |
| Figure 2.1.3-2 Worksheet | 9 |
| Figure 2.2.0-1 Display of a Imported Model..... | 11 |
| Figure 2.2.1-1 Calculation Log or Dayfile | 12 |
| Figure 2.2.2-1 Print Report Preview Options | 13 |
| Figure 2.2.2-2 Printing Options | 13 |
| Figure 2.3.1-1 ISO for New Problem..... | 16 |
| Figure 2.3.2-1 Main Screen – Setup Options | 18 |
| Figure 2.3.2-2 Project Data | 19 |
| Figure 2.3.2-3 Modeling Default | 20 |
| Figure 2.3.2-4 Case Definition Data | 20 |
| Figure 2.3.3-1 Node 1000, Anchor Dialog, Type/Location Tab | 21 |
| Figure 2.3.3-2 Node 1000, Anchor Dialog, Initial Mvmt/Rot Tab..... | 22 |

| | |
|--|----|
| Figure 2.3.3-4 Node 1000, Anchor Dialog, Process Tab | 24 |
| Figure 2.3.3-5 Node 1000, Anchor Dialog, Code Compliance Tab | 24 |
| Figure 2.3.3-6 Node 1000, Anchor Graphic Display | 25 |
| Figure 2.3.3-7 Node 1010, Pipe Dialog, Pipe Data Tab | 26 |
| Figure 2.3.3-9 Node 1010, Pipe Graphic Display (rotated) | 27 |
| Figure 2.3.3-8 Node 1010, Pipe Dialog, Restraints Tab | 27 |
| Figure 2.3.3-10 Node1020, Branch Dialog, Branch Connection Tab | 28 |
| Figure 2.3.3-11 Node 1020, Branch Joint Graphic Display | 29 |
| Figure 2.3.3-12 Node 1030, Pipe Dialog, Pipe Data Tab..... | 30 |
| Figure 2.3.3-13 Node 1030, Anchor Dialog, Type Location Data Tab | 31 |
| Figure 2.3.3-15 Node 1040. Elbow Data Dialog, Elbow Data Tab | 32 |
| Figure 2.3.3-16 Node 1040, Elbow Data Graphic Display | 33 |
| Figure 2.3.3-17 Node 1050, Branch Dialog, Branch Connection Tab..... | 34 |
| Figure 2.3.3-19 Node 1050, Branch Connection Graphic Display | 35 |
| Figure 2.3.3-20 Node 1060, Valve Dialog, Valve Data Tab..... | 37 |
| Figure 2.3.3-21 Graphic of Node 1060, Valve Dialog, Valve Data Tab | 37 |
| Figure 2.3.3-22 Node 1070, Pipe Dialog, Pipe Data Tab..... | 38 |
| Figure 2.3.3-23 Node 1070, Anchor Dialog, Type Location Tab | 39 |
| Figure 2.3.3-24 Node 1070, Pipe & Anchor Data Graphic Display | 39 |
| Figure 2.3.3-24 Node 1070, Pipe & Anchor Data Graphic Display | 40 |
| Figure 2.3.3-25 Node 1080, Pipe Data Sheet, Pipe Data Tab | 40 |
| Figure 2.3.3-26 Node 1080, Anchor Data Sheet, Type Location Tab | 41 |
| Figure 2.3.3-27 Node 1080, Pipe & Anchor Data Graphic Display | 41 |
| Figure 2.3.4-1 Main Screen, Calculate Pull-Down Menu..... | 42 |
| Figure 2.3.5-1 Output Pull-Down Menu | 43 |
| Figure 2.3.5-2 Output Load Case Pull-Down Menu..... | 44 |
| Figure 2.3.5-3 Output Type Pull-Down Menu | 44 |
| Figure 2.3.5-4 Output Code Compliance Report..... | 45 |
| Figure 2.3.5-5 Output Graphic, Output Display Menu | 46 |
| Figure 2.3.5-7 Output Graphic Screen | 47 |
| Figure 2.3.5-8 Output Graphic – Deformed Shape..... | 47 |
| Figure 2.3.6-1 Output Pull-Down Menu | 49 |

| | |
|---|----|
| Figure 2.3.6-2 Report Print Menu..... | 50 |
| Figure 2.3.6-3 Report Print Preview Sample..... | 50 |

2 TUTORIAL

2.1 Getting Started

To create a TRIFLEX Windows Icon on your desktop, do the following:

1. Click on the **START** button in the lower left corner of your screen.
2. Highlight **Find** and click on **Files or Folders**.
3. Enter **TriflexWindows.exe** in the *Named* field; select all hard drives in the *Look in* field and click on Find Now. The default path is:

C:\Program Files\PipingSolutions\TriflexWindows

4. Right click on the TriflexWindows.exe file name
5. Highlight *Create Shortcut* and left click
6. Click **YES** to respond to the Windows Message to place the TRIFLEX Windows Icon on the desktop.

To execute TRIFLEX Windows, double click on the TRIFLEX Windows Icon on the desktop.

To open an Existing Piping Model, click on **FILE** and from the pop-up menu, select **OPEN**. From the path (c:\ProgramsFiles\PipingSolutions\TriflexWindows\Samples\Tutorial01), open **Tutorial01.DTA** file.

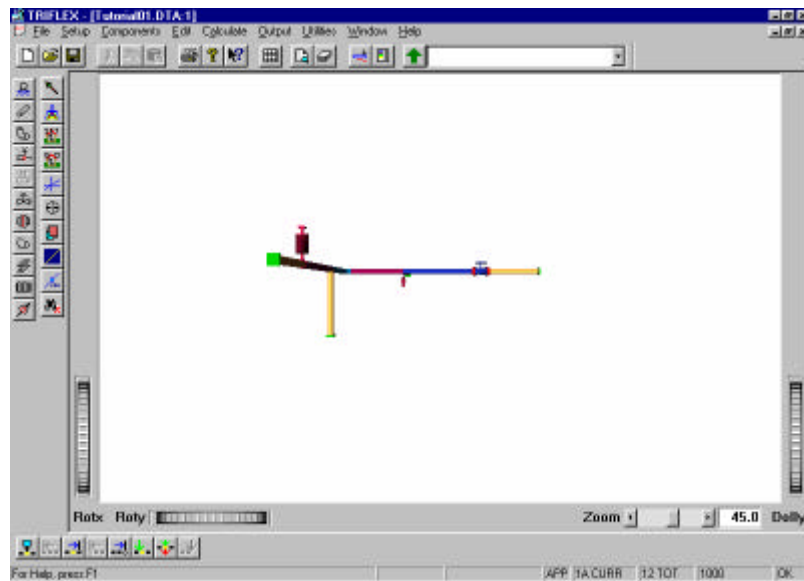
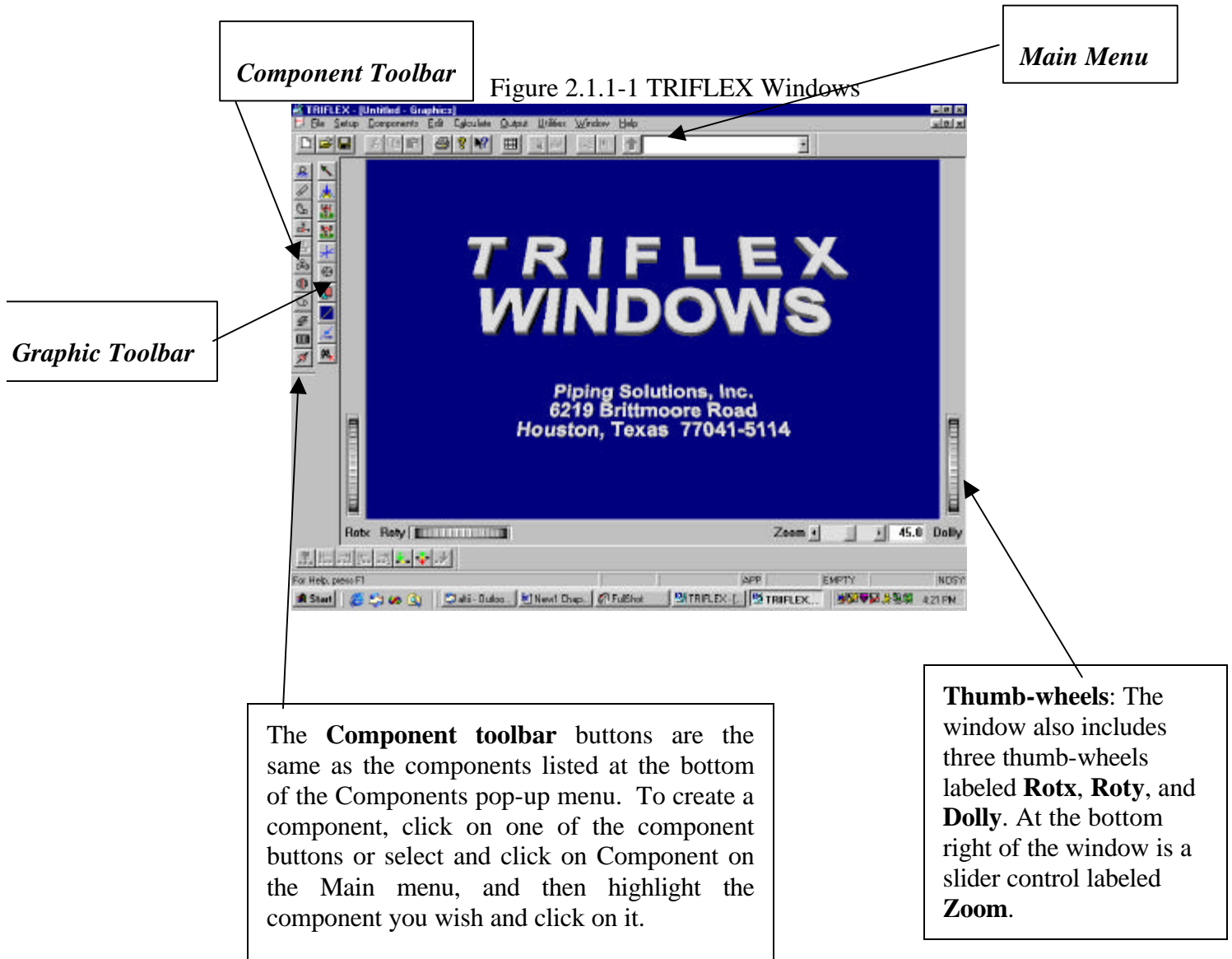


Figure 2.1.0-1 Demo IU1.dta Example

2.1.1 Main Screen Layout

When TRIFLEX is first brought up, the TRIFLEX introduction screen as shown in Figure 2.1.1-1 appears.



Graphics Window

Thumb-wheels & Zoom Slider

There are three wheels on the screen. The two thumb-wheels in the lower left corner: **Rotx**, **Roty** will rotate the piping system around x-axis and y-axis respectively. There is a third thumb wheel located on the lower right corner. In

Orthogonal mode, the thumb wheel will be labeled Zoom and will allow the user to zoom in and out on the model. In Perspective mode, the third thumb-wheel will be labeled Dolly and will enable the user zoom in and out in walk-through style. In Perspective mode, a slider is also provided in the lower right corner to enable the user to zoom in and out.

Note: +y axis is always up (vertical) in a piping model in TRIFLEX.

Toolbars and Menus

On left side of the screen, two Toolbars are provided. The buttons in the left column make up the Components Toolbar. The buttons in the right column make up the Graphic Toolbar.

Along the top of the screen, two rows of the Main Menu are provided. They are similar in style to the standard Microsoft Menu Layout and provide editing facilities, file services, graphic facilities, etc.

Figure 2.1.1-2 Status Bar Indicator



Status Bar

This is located on the bottom view of the screen (Figure 2.1.1-2).

APP - Refers to Append Mode as opposed to INS (Insert) Mode.

EMPTY – Appears when a piping model has not yet been created or loaded.

When a piping model has been created or loaded, the following two items will appear:









3B CURR- Current Component is No. 3 and is a Branch from node **1010** to **1020**.

12 TOT- Refers to the piping model having a total of 12 Components.

NOSYS - Appears when a piping model has not been created or loaded. When a piping model has been created or loaded, the following two items will appear to indicate the status of the geometry of the system:

OK- indicates that there is no geometry error.

ERR – indicates that there is a geometry error.

| | | | |
|---|------------------------|---|------------------|
|  | Edit current component |  | Last component |
|  | Previous component |  | Insert ahead |
|  | Next component |  | Replace current |
|  | First component |  | Append following |

2.1.2 Commands for Graphical Operations

Execute the following commands to become familiar with the Graphic Toolbar. We suggest you start with the Node Labels icon and work up to the Select/View icon, Figure 2.1.2-1.











| | |
|---|---|
|  | Select/View – Arrow used to point at a component and select it / Hand used to move or rotate the piping model |
|  | View All - Brings entire piping model into view on screen |
|  | Set Home – Allows user to define a view of the piping model as the default view |
|  | Go Home – Brings default view on screen |
|  | Toggle Axis – Draws X, Y, Z axis - size and position can be changed |
|  | Zoom Point – Brings user specified point in the piping model closer |
|  | Ortho/Perspective – Right angle view or panorama view |
|  | Line/Render – Line or 3D shapes –component colors can be changed |
|  | Node Labels – Node number on model – font size can be changed |
|  | Node Locate |

Figure 2.1.2-1 Graphic Toolbar Buttons

2.1.3 Accessing Data from Piping Model

To investigate the properties of a piping model, clicking (left mouse button) on the particular component of interest. For instance, clicking on the Anchor will

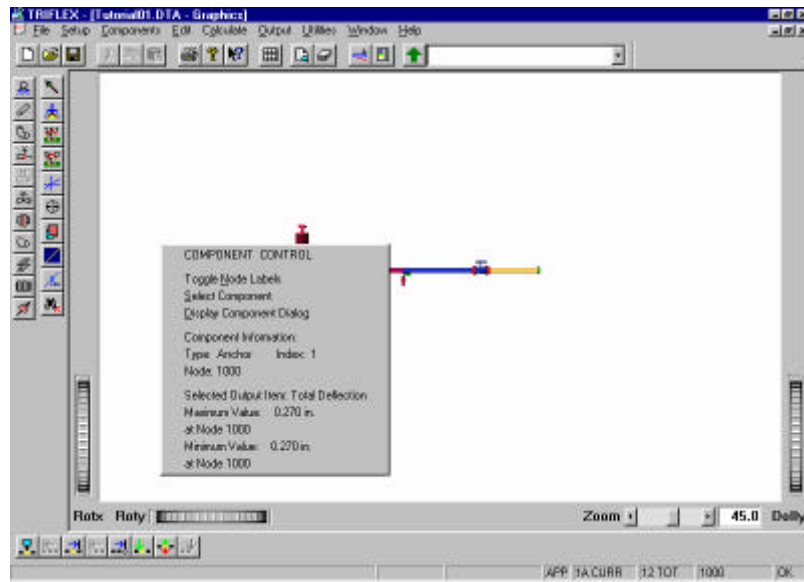


Figure 2.1.3-1 Viewing Anchor Component Properties

The screenshot shows the TRIFLEX spreadsheet window. It contains a table with 13 rows of component data. The columns are: Components, From, To, Delta X, Delta Y, Delta Z, Size, and Restraint. The data is as follows:

| | Components | From | To | Delta X | Delta Y | Delta Z | Size | Restraint |
|----|------------|------|------|---------------|---------|--------------|------|-----------|
| 1 | ANCHOR | 1000 | | 0 | 0 | 0.4 | | |
| 2 | PIPE | 1000 | 1010 | 0 | 0' | -4'-2.4 | | Rest |
| 3 | BRANCH | 1010 | 1020 | 0' | 0' | -6'-4" 4 | | |
| 4 | PIPE | 1020 | 1030 | 0' 3'-11-1/4" | 0' | 0.4 | | |
| 5 | ANCHOR | 1030 | | 0 | 0 | 0.4 | | |
| 6 | ELBOW | 1020 | 1040 | 0 | 0' | -4'-3-3/16.4 | | |
| 7 | BRANCH | 1040 | 1050 | 4' | 0' | 0.4 | | Rest |
| 8 | VALVE | 1050 | 1060 | 5'-10-1/8" | 0' | 0.4 | | |
| 9 | PIPE | 1060 | 1070 | 3'-3-3/8" | 0' | 0.4 | | |
| 10 | ANCHOR | 1070 | | 0 | 0 | 0.4 | | |
| 11 | PIPE | 1050 | 1080 | 0 | 0' | -5'-2-1/8.4 | | |
| 12 | ANCHOR | 1080 | | 0 | 0 | 0.4 | | |
| 13 | | | | | | | | |

Figure 2.1.3-2 Worksheet

yield a menu such as shown in Figure 2.1.3-1. To modify any property on this component, click on **Display Component Dialog** and enter the desired data in the

component dialog from the keyboard. An in-depth discussion can be found in Section 2.3.0.

To view entered data for the piping model, including node numbers, delta dimensions, pipe sizes, restraint indicators, pipe material, insulation material, and temperature and pressure for all load cases, click on the component button icon **Worksheet**, located in the Main Menu. Figure 2.1.3-2. Pressing the Ctrl + Tab keys allows the user to toggle between different screens.

Note: If your Company runs CAD from this system, then check to see what commands are “Hot Keyed”.

2.1.4 Using the Manual and Help Command

To access assistance with specific topics, click on **Help** on the Main Menu. **Index** and **User Manual** will then appear. Clicking on **Index** will show a list of topics to select from to obtain more detail about any specific topic listed. Clicking on **User Manual** will show a list of the chapters available for viewing.

The electronic TRIFLEX User’s Manual is located in the default directory:

c:\ProgramFiles\PipingSolutions\TriflexWindows\Manual

The manual is furnished electronically in Adobe Acrobat (*.pdf) format and linked by chapter, figures and index. Click on a chapter and the chapter will appear on the screen.

2.2 Opening and Importing Example Piping Model Files

To open a previously created piping model, click on **File** in the **Main Menu**, select option **Open** and then select the file you wish. By default, the extension of TRIFLEX data files is “.dta”. The complete path is:

c:\ProgramFiles\PipingSolutions\TriflexWindows\Samples\Tutorial01\Tutorial01.dta

To import a previously created TRIFLEX DOS piping model, click on **Utilities** in the **Main Menu**, select option **Import File** and then click on **DOS Triflex Job**. By default, the extension of DOS TRIFLEX data files is “.job”. The complete path is:

**c:\Programs
Files\PipingSolutions\TriflexWindows\Samples\Tutorial01\Tutorial01.job**

To display the spreadsheet and the piping model simultaneously on a split screen as shown in Figure 2.2.0-1, open a piping model. The piping model will be displayed on the screen. Click on **Windows** on the **Main Menu** and select **Tile Vertical**. The user will see two windows; one with the piping model and the other will be blank. The user should then click on the Spreadsheet Icon in the Main Menu to obtain the spreadsheet in the blank screen. Click on any component in the piping model and the data for that component will be highlighted in the spreadsheet. Similarly, by clicking on a node in the spreadsheet, the component on the piping model will be highlighted. This is useful in identifying components in a piping model for copying, inserting and deleting.

Note: Models may be built using the spreadsheet and/or in graphic mode as described in section 2.3.0 of this User's Manual.

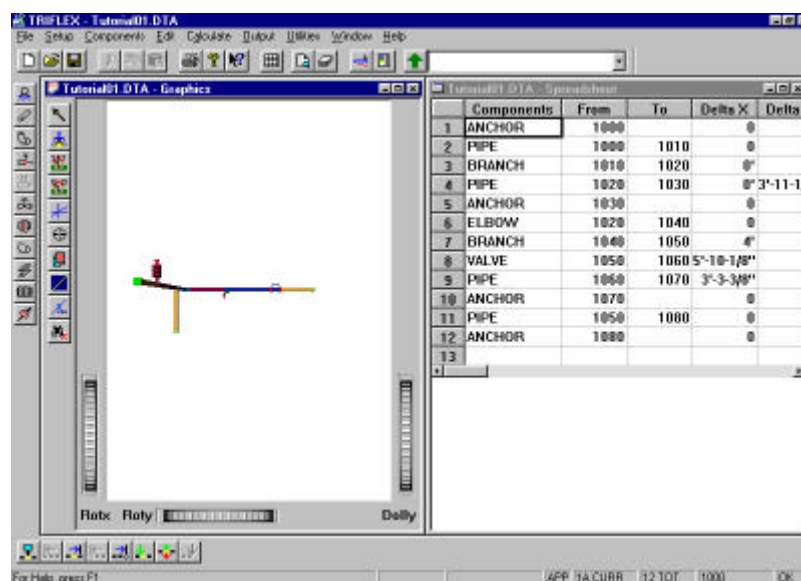


Figure 2.2.0-1 Display of a Imported Model

Note: In order to **PAN** hold down the SHIFT key and left Click on the mouse on the model dragging the chosen area of the model to the center position Appendix A lists Keyboard Control Key.

2.2.1 Processing a Previously Built Piping Model

There are two methods of processing a piping stress analysis. The first is to go to the **Main Menu**, select **Calculate** and then select **Basic Calculation**. The second method is to click on the green arrow on the **Main Toolbar**. Figure 2.2.1-1 depicts the “run time log” or “calculation log” sometimes known as the “Dayfile”. While the program is executing or after the program has executed, the user should look for the following terms: *ERROR; QUIT; EXIT: Normal Termination*

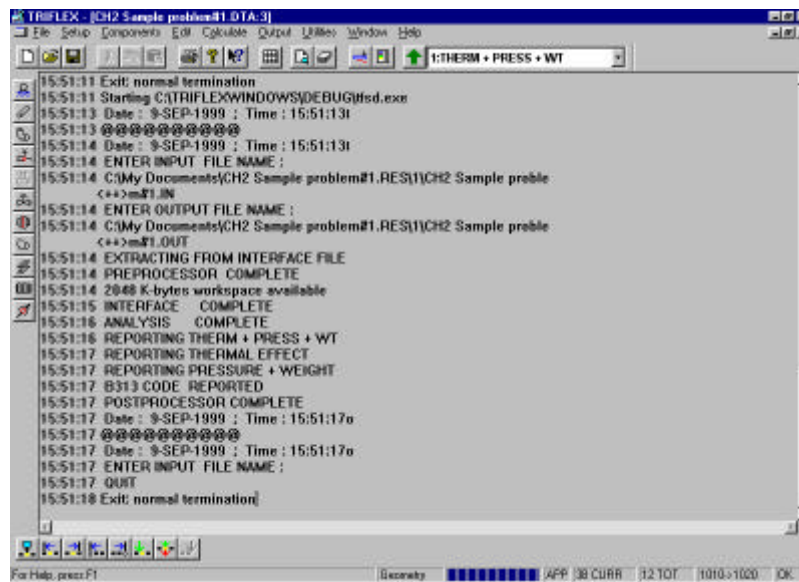


Figure 2.2.1-1 Calculation Log or Dayfile

If the last two lines of the dayfile are “QUIT” and “Exit: normal termination”, then TRIFLEX Windows is telling you that the execution was successfully completed. If the word “**ERROR**” appears, then you must examine your input data to find the error and make corrections. Please note that TRIFLEX generates this report in another window for viewing. To return to the piping model, you must delete or minimize this window.

NOTE: If you have imported a DOS TRIFLEX data file, you must re-define the required case data. To do so, Click on **Setup** on the **Main Menu** and then click on the **Case Definition**. The user must enter the desired case data on the dialog provided.

2.2.2 Printing Output Reports

To print output reports, click on **Output** on the **Main Menu** and then click on **Print Reports** on the **Pulldown Menu**. The screen in Figure 2.2.2-1 will appear.

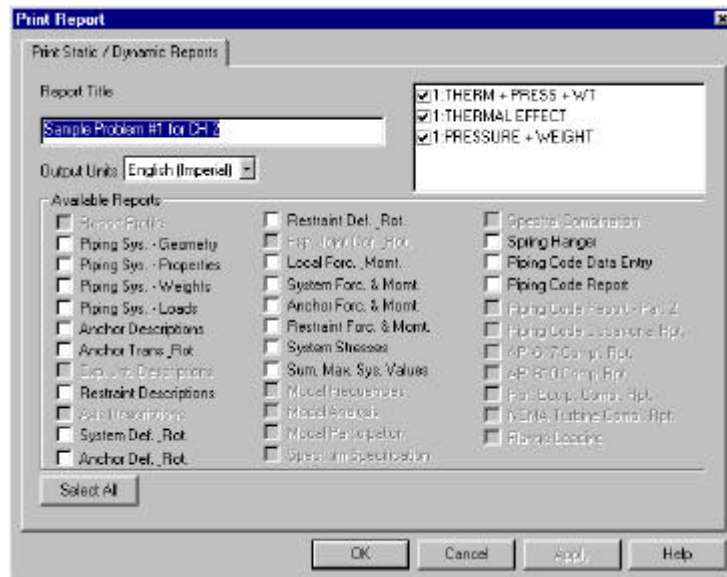


Figure 2.2.2-1 Print Report Preview Options

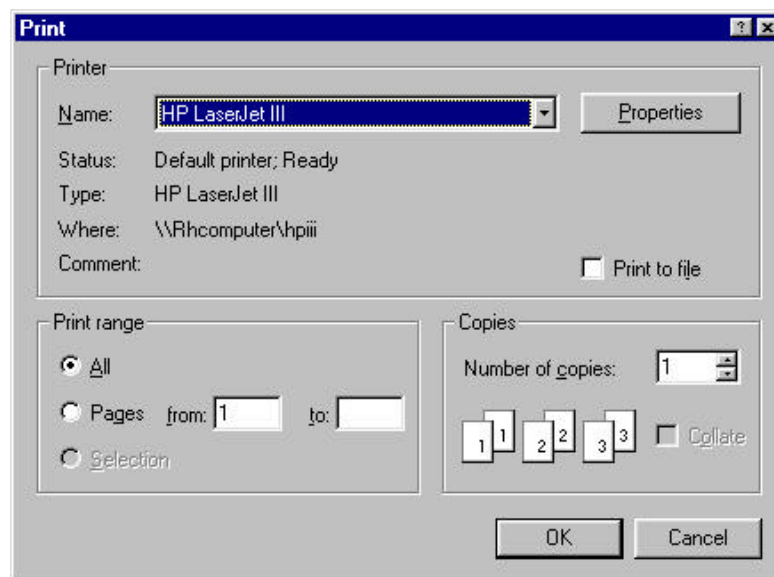


Figure 2.2.2-2 Printing Options

Select the desired load cases and check the reports you wish to review and click the OK button. TRIFLEX will then give you an opportunity to select the printer and printing options as shown in Figure 2.2.2-2 and will then print the reports for you.

2.2.3 Append, Insert and Replace Mode

In order to demonstrate the modification capabilities of TRIFLEX Windows, it is best to either create a short model or refer to Figure 2.1.0-1. TRIFLEX Windows can operate in **APPEND** mode, **INSERT** mode or **REPLACE** mode. To change this mode, click on **Components** on the **Main Menu** and then click on the desired mode - Append, Insert or Replace. Alternatively, the user can click on the icons located in the bottom left corner of Main Screen to change the operating mode. See Figure 2.1.2-2 for an explanation of these Icons.

The three modes for modeling a component are as follows: **Insert** (creates component prior to highlighted or current component), **Append** (creates component following last component in a branch) and **Replace** (replaces highlighted or current component). When building a new piping model, the user must be in Append mode. When the user wishes to insert a new component in an existing piping model prior to a highlighted component, the Insert mode should be selected. When the user wishes to replace one highlighted component, the user should select the replace mode. Insert and Replace also are functional for current or last coded components when no component is highlighted. The selected mode will remain the same until the user selects a different mode.

To **Insert** one or more components, do the following:

1. Turn on the node numbers by clicking on the **Node Numbers** Icon on the Graphic Toolbar while viewing the piping model.
2. Highlight the component before which you wish to place a new component. Alternatively, you can select this component on the spreadsheet.
3. Click on the **Insert** Icon in the lower left corner.
4. Select the component you wish to insert from the component toolbar and the desired dialog will appear for you to define the component. Then click **OK** or press **Enter**.

Similarly, to **Append** a component following **the last component** (must be last component of a branch), click on the desired component on the component toolbar and enter the data on the dialog that appears. Then click **OK** or press **Enter**.

To **Replace** a component, do the following:

1. Turn on the node numbers by clicking on the **Node Numbers** Icon on the Graphic Toolbar while viewing the piping model.

2. Highlight the component which you wish to replace. Alternatively, you can select this component on the spreadsheet.
3. Click on the **Replace** Icon in the lower left corner.
4. Select the new component from the component toolbar. The desired dialog will appear for you to define the component. Then click **OK** or press **Enter**.

Modifying (Delete, Cut, Paste, Copy and Undo)

The following procedures are recommended for graphically modifying components:

Deleting

1. Click on the component(s) to be deleted.
2. Press the **Del (Delete)** key.

Cutting (Ctrl + x)

1. Click on the component(s) that are to be cut.
2. Click on **Edit** on the **Main Menu** and click on **Cut**.

Copying (Ctrl + c)

1. Click on the component(s) that are to be copied.
2. Click on **Edit** on the **Main Menu** and click on **Copy**.

Pasting (Ctrl + v) May be used to append one or more components (previously cut or copied components) to the TO node of the highlighted component.

1. Click on the component to which the component(s) are to be pasted.
2. Click on **Edit** on the **Main Menu** and click on **Paste**.

Undo (Ctrl +z) To undo the last operation, click on **Edit** on the **Main Menu** and click on **Undo**.

Note: Appendix A lists Keyboard Control Key.

2.3 Coding the New Sample Problem

The purpose of this section is to demonstrate the entry of data into the TRIFLEX Windows dialogs and to build a small piping model.

A piping model will be generated using the interactive screen capabilities. This model will illustrate a portion of the TRIFLEX Windows features and will provide a solid basis for utilizing all of the TRIFLEX Windows capabilities.

2.3.1 Define the Problem

Objective is to run a static operating case analysis for the piping system shown in Figure 2.3.1-1.

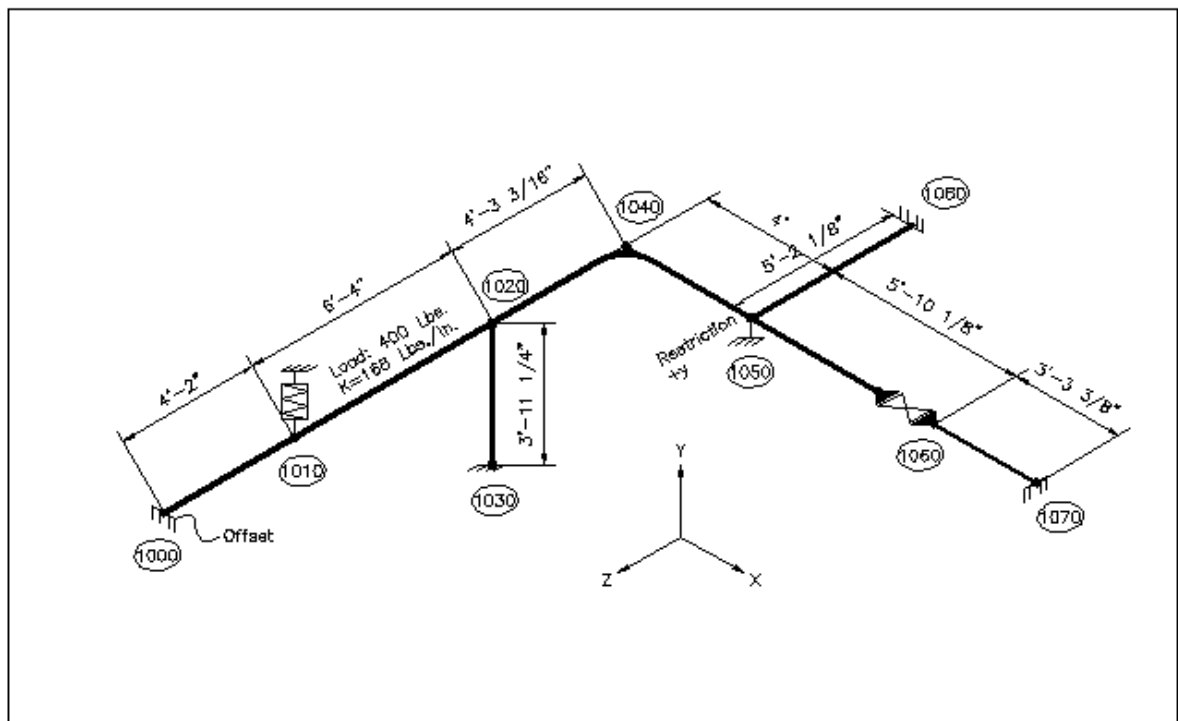


Figure 2.3.1-1 ISO for New Problem

Engineering Information regarding Model to be built.

Nominal Pipe Diameter = 4 inch

Schedule = Standard Wall

Pipe Material = Carbon Steel

Modulus of Elasticity = Installed

Insulation Material = Calcium Silicate

Insulation Thickness = 2 inches

Contents Specific Gravity = 0.85

Design Pressure = 70 psig

Design Temperature = 300 °F

Offset dimension for first anchor in the y direction = 27'-2-3/4"

Temperature for Offset = 200 °F

Valve = Flange Gate Valve

Flanges = Weld Neck Flanges

Flange Class = 150

2.3.2 Starting Triflex Windows

Begin by double clicking on the TRIFLEX Windows icon on your desktop.

After the logo screen appears for a few seconds, the main screen of TRIFLEX Windows will be displayed.

1. From **Setup** menu, select **Project** as shown in Figure 2.3.2-1

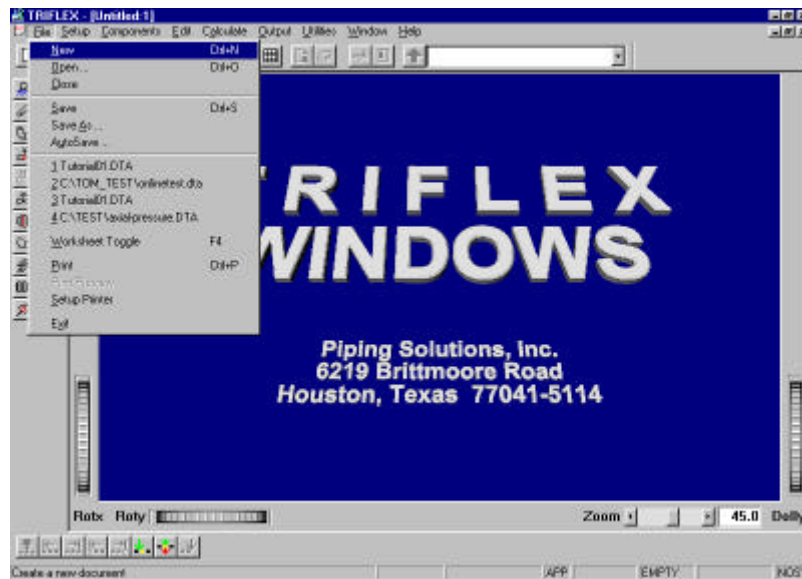
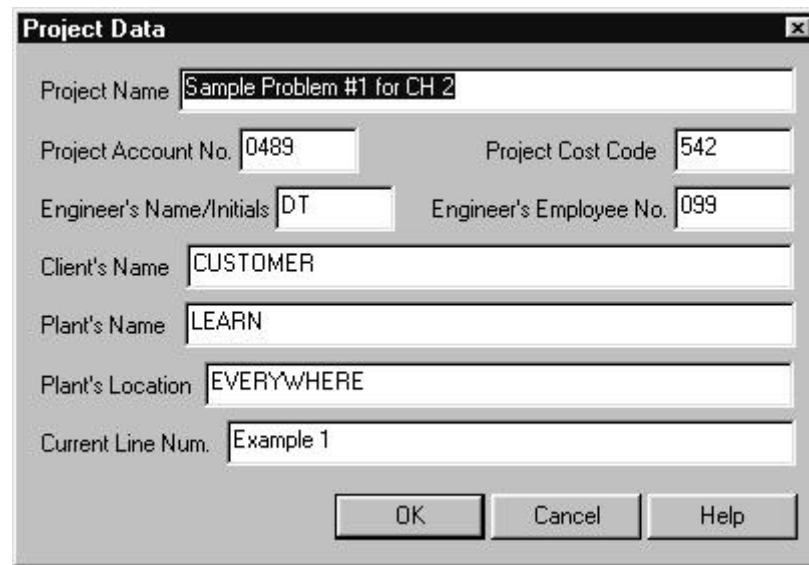


Figure 2.3.2-1 Main Screen – Setup Options

Complete the fields to define Project Name, Project Account No., Project Cost Code, Engineer's Name/Initials, etc., as shown in Figure 2.3.2-2. These fields are not mandatory to execute an analysis of the above model.



Project Data

Project Name: Sample Problem #1 for CH 2

Project Account No.: 0489 Project Cost Code: 542

Engineer's Name/Initials: DT Engineer's Employee No.: 099

Client's Name: CUSTOMER

Plant's Name: LEARN

Plant's Location: EVERYWHERE

Current Line Num.: Example 1

OK Cancel Help

Figure 2.3.2-2 Project Data

2. From the **Setup** menu, select **Modeling Defaults** and the screen shown in Figure 2.3.2-3 will appear.

Use the mouse or tab key to move to **Initial Node Number** field and enter **1000**.

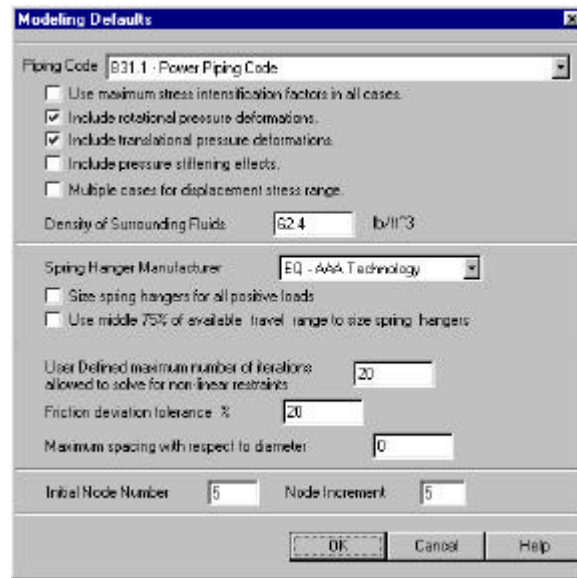
Use the mouse or tab key to move to **Node Increment** field and enter **10**.

Accept the other defaults set by TRIFLEX on this screen.

3. From **Setup** menu, select **Case Definition Data** and the screen shown in Figure 2.3.2-4 will appear.

In **Case#1**, use the mouse to place a check in the box by **Process this Case**.

In **Case#1**, use the mouse to place a check in the box by **Perform Operating Case Analysis**.



Modeling Defaults

Piping Code: **B31.1 - Power Piping Code**

☐ Use maximum stress intensification factors in all cases.
☒ Include rotational pressure deformations.
☒ Include translational pressure deformations.
☐ Include pressure stiffening effects.
☐ Multiple cases for displacement stress range.

Density of Surrounding Fluids: **62.4** lb/ft³

Spring Hanger Manufacturer: **EQ - A&A Technology**

☐ Size spring hangers for all positive loads.
☐ Use middle 75% of available travel range to size spring hangers.

User Defined maximum number of iterations allowed to solve for non-linear restraints: **20**

Friction deviation tolerance %: **20**

Maximum spacing with respect to diameter: **0**

Initial Node Number: **5** Node Increment: **5**

OK Cancel Help

Figure 2.3.2-3 Modeling Default



Case Definition Data

| LOAD CASE# | #1 | #2 | #3 | #4 | #5 | #6 |
|---------------------------------|-------------------------------------|--------------------------|--------------------------|--------------------------|--------------------------|--------------------------|
| Process this Case | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Perform Operating Case Analysis | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Perform Hydro-test Case(P+W/t) | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Perform Piping Code Compl. | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Temperature | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Pressure | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Weight | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Anchor Movements | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Restraint Movements | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Restraint Loads | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Contents Weight | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Insulation | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Soil Interaction | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Wind Only Loading as Load | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Wind Loads as Occasional | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Seismic Loads - Static | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Mode Shapes and Frequencies | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |

OK Cancel Help

Figure 2.3.2-4 Case Definition Data

2.3.3 Coding the Components

2.3.3.1 Anchor Data Point 1000

To specify the anchor at data point **1000**, you may either click on **Components** on the Main Menu and select the **Anchor** Menu Item, or click on the convenient **Anchor** Icon on the **Component** toolbar. The Anchor dialogs will be displayed as shown in Figure 2.3.3-1. All of the input fields on the tabbed Anchor dialogs are for the specification of anchor properties, movements and stiffness (translational and/or rotational).

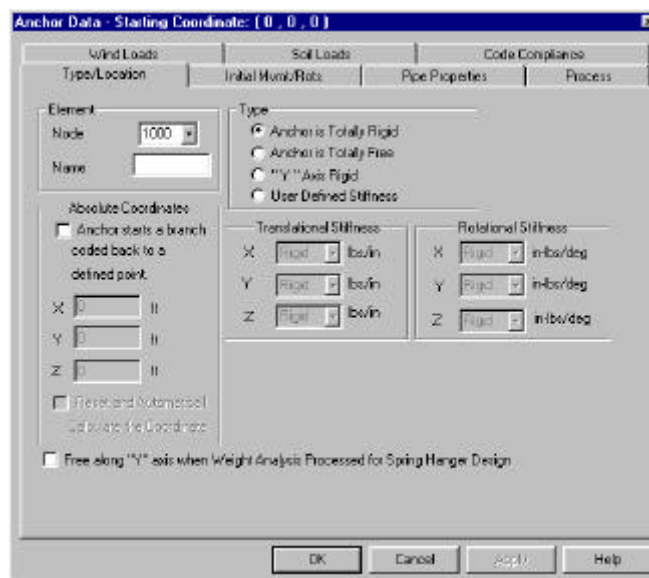


Figure 2.3.3-1 Node 1000, Anchor Dialog, Type/Location Tab

The anchor at data point **1000** was selected as a **Totally Rigid Anchor** and this allows the piping system to grow vertically due to the coefficient of expansion of the vessel where it was attached. To view other Anchor dialogs:

1. Select **Initial Mvmt/Rots** tab – Figure 2.3.3-2
2. Select material for Offset (default is **LS – Low Carbon Steel**), Temperature of Offset (in our case 200 °F) and type in Offset at a distance of 27 feet, 6 inches (this is typed in as **27-6**). This distance is the actual distance from the true anchor to the point where the piping system components begin. TRIFLEX® Windows will calculate the vertical growth for data point **1000** from the true anchor point to the start of the piping system as shown in Figure 2.3.3-2.

Anchor Data

Wind Loads Soil Loads Code Compliance

Type/Location Initial Mvmt/Rot Pipe Properties Process

Initial Movements

| Case | #1 | #2 | #3 | #4 | #5 | #6 | |
|------|----------|----|----|----|----|----|----|
| X | 0 | 0 | 0 | 0 | 0 | 0 | in |
| Y | 0.269569 | 0 | 0 | 0 | 0 | 0 | in |
| Z | 0 | 0 | 0 | 0 | 0 | 0 | in |

Initial Rotations

| | #1 | #2 | #3 | #4 | #5 | #6 | |
|---|----|----|----|----|----|----|-----|
| X | 0 | 0 | 0 | 0 | 0 | 0 | Deg |
| Y | 0 | 0 | 0 | 0 | 0 | 0 | Deg |
| Z | 0 | 0 | 0 | 0 | 0 | 0 | Deg |

Offset Material, Dimensions and Temperatures

Material: LS-Low Carbon Steel (c 0.3% C)

Temp: 60 F

Offset Dimensions: X: 0, Y: 272-3/4, Z: 0

OK Cancel Apply Help

Figure 2.3.3-2 Node 1000, Anchor Dialog, Initial Mvmt/Rot Tab

3. Select the **Pipe Properties** Tab as shown in Figure 2.3.3-3.

Use the tab key or the mouse to move to the **Nominal Diameter** field of the **Pipe Size** field group. Select **4"** from that pull down menu. Notice that the **Outside Diameter** field is automatically filled in after you move from the **Nominal Diameter** field. TRIFLEX® Windows has a complete set of piping data stored and it automatically retrieves information about the outside diameter for a pipe size with a nominal diameter of 4 inches. Use the tab key or mouse to select the **Pipe Schedule** field. Select **Standard (STD)** from that pull down menu.

Use the tab key or your mouse to move to the **Material** field of the **Pipe Material** field group. Use the pull down menu to select the **Pipe Material**.

Note: The pull down menu will list all material codes that TRIFLEX® Windows has stored in its internal database. The internal TRIFLEX® Windows database may be viewed by the user in Access (look for the **Find: triflex.mdb** file). This database is listed as: a flange, insulated material, material1, material2, pipe, structural steel, and valve database with the appropriate information under each one.

Figure 2.3.3-3 Node 1000, Anchor Dialog, Pipe Properties Tab

For the purpose of this example, **Low Carbon Steel (LS)** should be selected. Select carbon steel as the pipe material and note that the pipe density appears. Notice that when the pipe density appears, TRIFLEX® Windows will calculate the weight per unit length of the pipe and show it in the appropriate field. If this weight does not agree with the weight in your specifications, you may select the **User Defined** Material to change the **Density** value.

Use the tab key or the mouse to move to the **Material** field of the **Insulation** field group. Use the pull down menu to select the insulation **Material** for the **Pipe**. Use the pull down menu to select **(CS) Calcium Silicate** as the insulation material. Enter 2 in the insulation thickness field. Notice the calculation of weight is per unit length. If this weight does not agree with the weight in your specifications, select the **User Defined** Material to change **Density** value.

4. Select the **Process** Tab to enter the pressure and temperature of the piping system as shown in Figure 2.3.3-4.
5. Use the tab key or your mouse to move to the **Base Temperature** field and enter **70 °F**.
6. Use the tab key or your mouse to move to the **Pressure** field and enter **70** psig.
7. Use the tab key or your mouse to move to the **Temperature** field and enter **300 °F**.
8. Select the **Code Compliance** Tab to enter the Stress Allowable Values and related data as shown in Figure 2.3.3-5

Anchor Data - Starting Coordinate: (0 , 0 , 0)

Wind Loads Soil Loads Code Compliance

Type/Location Initial Mom/Rots Pipe Properties Process

Material

Material:

Base Temperature: F

| Case: | #1 | #2 | #3 | #4 | #5 | #6 |
|------------------------|-----------------------------------|-----------------------------------|-----------------------------------|-----------------------------------|-----------------------------------|-----------------------------------|
| Pressure: | <input type="text" value="70"/> | <input type="text" value="0"/> | <input type="text" value="0"/> | <input type="text" value="0"/> | <input type="text" value="0"/> | <input type="text" value="0"/> |
| Temperature: | <input type="text" value="300"/> | <input type="text" value="70"/> | <input type="text" value="70"/> | <input type="text" value="70"/> | <input type="text" value="70"/> | <input type="text" value="70"/> |
| Use Installed Modulus: | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Use Operating Modulus: | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Modulus of Elasticity: | <input type="text" value="29.5"/> | <input type="text" value="29.5"/> | <input type="text" value="29.5"/> | <input type="text" value="29.5"/> | <input type="text" value="29.5"/> | <input type="text" value="29.5"/> |
| Coeff of Exp: | <input type="text" value="1.82"/> | <input type="text" value="0"/> | <input type="text" value="0"/> | <input type="text" value="0"/> | <input type="text" value="0"/> | <input type="text" value="0"/> |

Units: m-PSI in/100ft

Figure 2.3.3-4 Node 1000, Anchor Dialog, Process Tab

Anchor Data

Type/Location Initial Mom/Rots Pipe Properties Process

Wind Loads Soil Loads Code Compliance

B31.3 Refinery Piping Code Compliance Data

Conservative Allowables, C: ☐

Joint Factor, E:

Allowable Cold Stress, SC: psi

Allowable Hot Stress, SH1: psi

Allowable Hot Stress, SH2: psi

Allowable Hot Stress, SH3: psi

Allowable Hot Stress, SH4: psi

Allowable Hot Stress, SH5: psi

Allowable Hot Stress, SH6: psi

Stress Range Reduction Factor, F:

Coefficient in Code Book, Y:

Occasional Load Factor, K:

Mill Tolerance, M: % or in

Figure 2.3.3-5 Node 1000, Anchor Dialog, Code Compliance Tab

- 9 Click the **OK** or press the **Enter** key on the Anchor dialogs to save the anchor data. An anchor will be displayed in the graphic window as shown in Figure 2.3.3-6

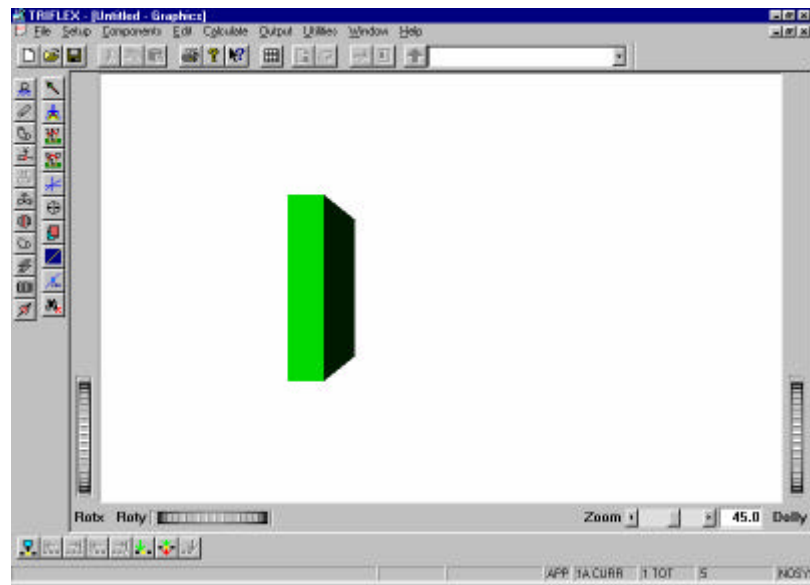


Figure 2.3.3-6 Node 1000, Anchor Graphic Display

2.3.3.2 Pipe Data Point 1010

To specify the pipe component, do the following

1. Select the convenient **Pipe** Icon from the **Component** toolbar.
2. Tab to or use the mouse to move to the **Delta Z** field of the **Dimensions from “From Node” to “To Node”** field group. Enter **–4’-2”** in the **Delta Z** field to indicate that the pipe segment is 4 foot 2 inches long and that the direction from the **From Node** to the **To Node** is in the negative Z direction as shown in Figure 2.3.3-7.
3. Select **Restraint** Tab as shown in Figure 2.3.3-8.

The screenshot shows the 'Pipe Data' dialog box with the 'Restraints' tab selected. The 'Element' section shows 'From Node' as 1000 and 'To Node' as 1010. The 'Dimensions from "From Node" to "To Node"' section shows Delta X as 0 ft, Delta Y as 0 ft, Delta Z as -4'2" ft, and Abs Length as 4'2" ft. The 'Number of Intermediate Nodes' is 0 and 'Max. Spacing' is 0 ft. The 'Cold Spring' section has 'Cut Short' and 'Cut Long' checkboxes. The 'Stress Intensification Factor' section has fields for 'for "From Node"' and 'for "To Node"'. The 'Weight Off' and 'Buoyancy calculations' checkboxes are unchecked. The 'Pipe Size' section shows 'Nominal Diam' as 4 in and 'Pipe Sch' as STD. A note at the bottom states 'Pipe size can be changed only from Properties Screen'.

Figure 2.3.3-7 Node 1010, Pipe Dialog, Pipe Data Tab

4. In the **Spring Hanger** field group, click on the **Existing Spring Hanger** check box.
5. Enter **400** lbs in **Installed Load** field and **168** lbs/in. in the **Spring Rate** field as shown in Figure 2.3.3-8.

Click **OK** to save the data and to display the Graphic View shown in Figure 2.3.3-9.

Pipe Data

Tab: Restraints

Element: Load Case: Node: 1010 Name:

Coordinate System: ☒ X, Y, Z Coord. System ☐ L, N, G Coord. System ☐ A, B, C Coord. System

☐ Use Direction Vectors ☒ Use Spring Angles

Action Angles: X-Axis: Y-Axis: Z-Axis:

A-Angle: B-Angle: C-Angle:

Friction Coefficient: 0

Translational Restraint Action

| X Axis | Y Axis | Z Axis |
|-----------------|-----------------|-----------------|
| + and - | + and - | + and - |
| Limit Stops | Limit Stops | Limit Stops |
| MOV: in | MOV: in | MOV: in |
| Force: lbs | Force: lbs | Force: lbs |
| Stiff: lbs/in | Stiff: lbs/in | Stiff: lbs/in |
| Damp: in-lbs/in | Damp: in-lbs/in | Damp: in-lbs/in |

Rotational Restraint Action

| X Axis | Y Axis | Z Axis |
|-------------------|-------------------|-------------------|
| + and - | + and - | + and - |
| ROT: deg | ROT: deg | ROT: deg |
| Mom: ft-lbs | Mom: ft-lbs | Mom: ft-lbs |
| Stiff: in-lbs/deg | Stiff: in-lbs/deg | Stiff: in-lbs/deg |

Spring Hanger

☐ Size a Spring Hanger ☒ Existing Spring Hanger

Allowed Load Variation: % Installed Load: 400 lbs

No. of Spring Hanger: Spring Rate: 160 lbs/in

Buttons: OK, Cancel, Apply, Help

Figure 2.3.3-9 Node 1010, Pipe Graphic Display (rotated)

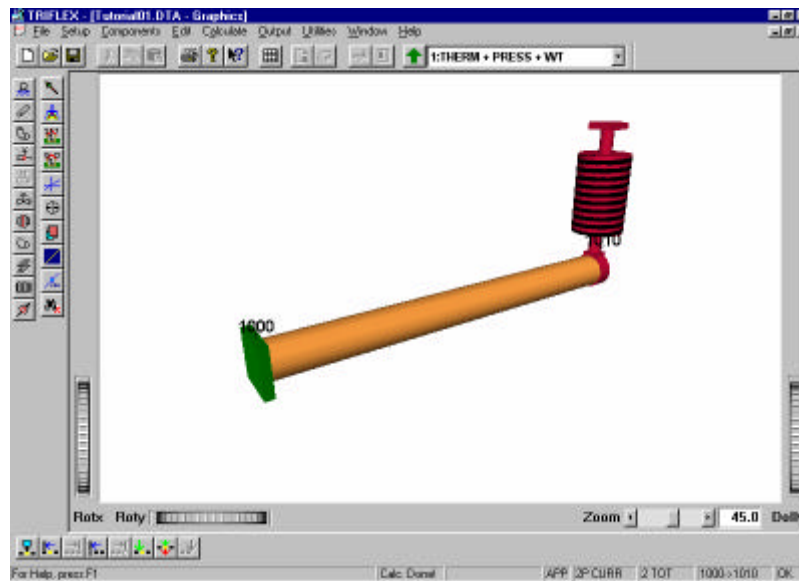


Figure 2.3.3-8 Node 1010, Pipe Dialog, Restraints Tab

2.3.3.3 Branch Connection Data Point 1020

1. Click the **Branch Connection** Icon on the **Components** toolbar to display the **Branch Connection Data** dialog as shown in Figure 2.3.3-10. The node numbers of the node points are assigned by TRIFLEX automatically. If the user wishes a different labeling of the node points for the Branch Connection, use the pull down menu to select the node number or type in the node number you wish.

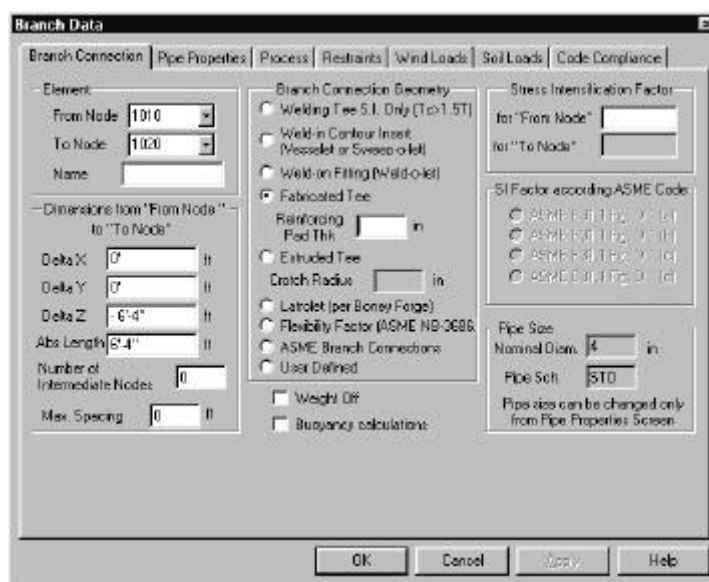


Figure 2.3.3-10 Node1020, Branch Dialog, Branch Connection Tab

2. Select the **Fabricated Tee** radio button from Branch Connection Geometry group.
3. Tab to or use the mouse to move to the **Delta Z** field of the **Dimensions from “From Node” to “To Node”** field group. Enter **-6’-4”** in the **Delta Z** field to indicate that the pipe segment is 6 foot 4 inches long and that the direction from the **From Node** to the **To Node** is in the negative Z direction as shown in Figure 2.3.3-10.

Data Point 1020 is now defined as the midpoint of a Fabricated Tee Branch Connection and will be intensified accordingly. With the indication of the type of Branch Connection, TRIFLEX® Windows can calculate the proper stress intensification per the Piping Code specified on the **Modeling Default** dialog under the **Setup**.

To specify your own stress intensification factor at a branch connection, select the **User Defined** radio button from the **Branch Connection Geometry** group and you can enter the values you wish for the **From Node** and **To Node** fields in the **Stress Intensification Factor** group.

4. Click **OK** to save the pipe and branch connection data and display the Graphic View shown in Figure 2.3.3-11.

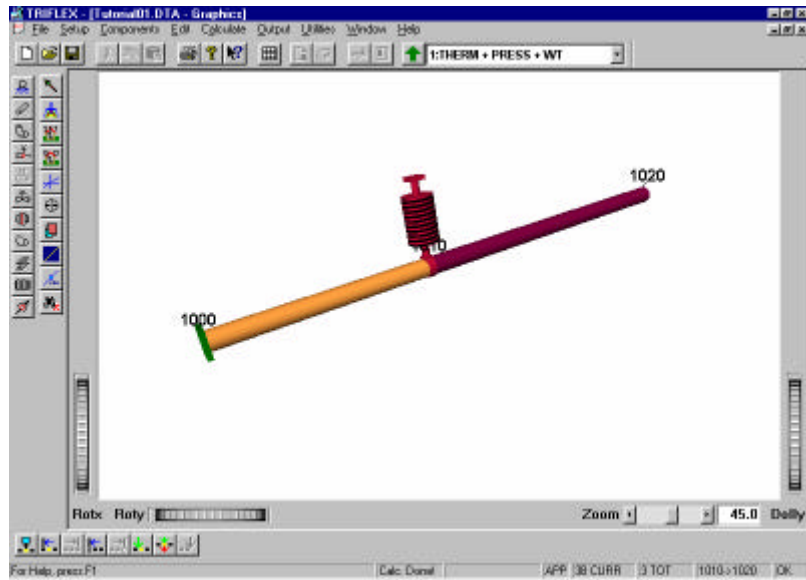


Figure 2.3.3-11 Node 1020, Branch Joint Graphic Display

2.3.3.4 Pipe & Anchor Data Point 1030

1. Click on the **Pipe** Icon on the **Component** toolbar.
2. Tab to or use the mouse to move to the **Delta Y** field of the **Dimensions from "From Node" to "To Node"** field group. Enter **-3'-11-1/4"** in the **Delta Y** field to indicate that the pipe segment is 3 foot 11 1/4 inches long and that the direction from the **From Node** to the **To Node** is in the negative Y direction as shown in Figure 2.3.3-12.
3. Click **OK** to save the pipe data.

Figure 2.3.3-12 Node 1030, Pipe Dialog, Pipe Data Tab

The screenshot shows the 'Pipe Data' dialog box with the following fields and values:

- Element:**
 - From Node: 1020
 - To Node: 1030
 - Name: (empty)
- Dimensions from "From Node" to "To Node":**
 - Delta X: 0 ft
 - Delta Y: -3'-11-1/4 in
 - Delta Z: 0 ft
 - Abs Length: 3'-11-1/4 in
 - Number of Intermediate Nodes: 0
 - Max Spacing: 0 ft
- Cold Spring:**
 - ☐ Cut Short
 - ☐ Cut Long
 - Cut Length: 0 in
- Stress Intensification Factor:**
 - for "From Node": (empty)
 - for "To Node": (empty)
- Weight OR Buoyancy calculations:**
 - ☐ Weight OR
 - ☐ Buoyancy calculations
- Pipe Size:**
 - Nominal Diam: 4 in
 - Pipe Sch: STD

At the bottom, there is a note: "Pipe size can be changed only from Properties Screen". The dialog has buttons for OK, Cancel, Apply, and Help.

4. Click on the **Anchor** Icon on the **Component** toolbar.
5. Verify that the number **1030** is in the **Node Label** field of the **Element** field group for the anchor as shown in Figure 2.3.3-13.
6. Click **OK** to save the anchor data and display the Graphic View shown in Figure 2.3.3-14.

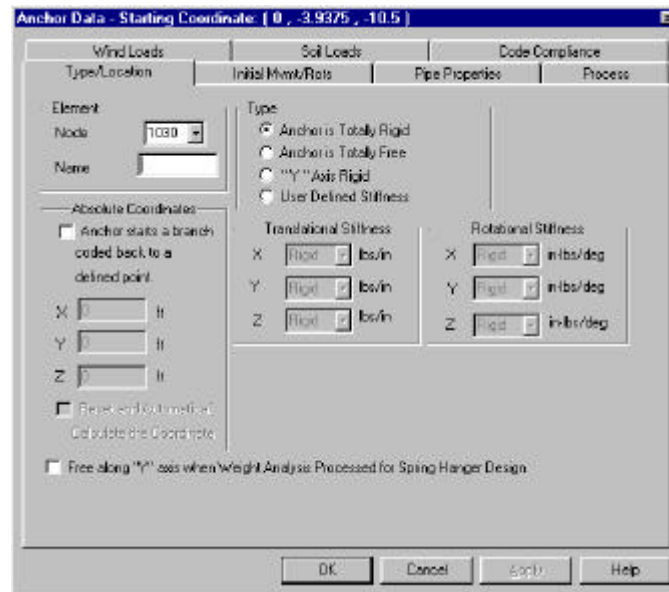


Figure 2.3.3-13 Node 1030, Anchor Dialog, Type Location Data Tab

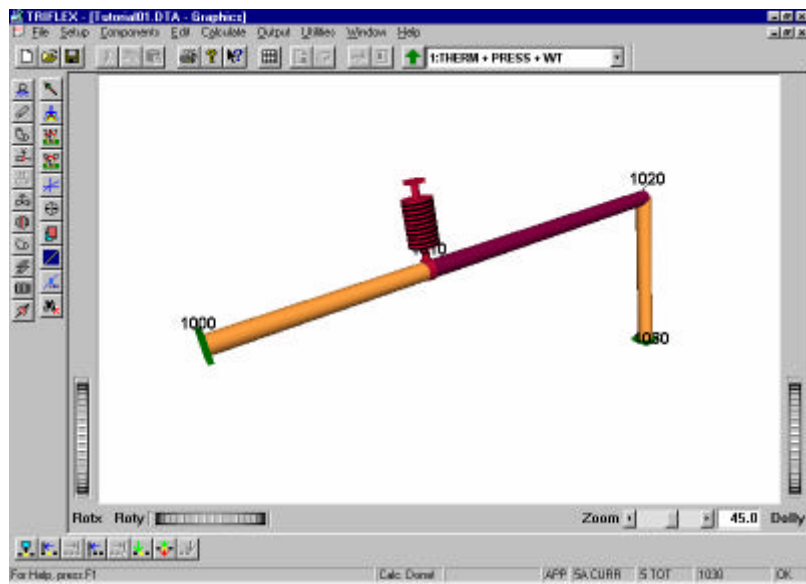


Figure 2.3.3-14 Node 1030, Pipe and Anchor Graphic Display

2.3.3.5 Elbow Data Point 1040

- 1 Click on the **Elbow** Icon on the **Component** toolbar to display the **Elbow Data** dialog as shown in Figure 2.3.3-15.
- 2 Use the pull down menus to select the correct node numbers for the **From Node - 1020** and the **Tangent Intersection Node - 1040**.
- 3 Tab to or use the mouse to move to the **Delta Z** field of the **Dimensions from "From Node" to "To Node"** field group. Enter **-4-3-3/16"** in the **Delta Z** field to indicate that the elbow and preceding pipe segment is 4 foot 3 3/16 inches long and that the direction from the **From Node** to the **To Node** is in the negative Z direction as shown in Figure 2.3.3-15.
- 4 Enter **4'** in the **Delta X** field of the **Dimension from "Tangent Intersection To Next Component"** group.

Figure 2.3.3-15 Node 1040. Elbow Data Dialog, Elbow Data Tab

Elbow Data

Elbow/Bend Element

From Node: 1020

Tangent Intersection Node: 1040

Name:

Elbow/Bend Properties

☒ Elbow ☐ Bend Filling Thick: 0.237 in

☒ Long Radius ☐ Short Radius ☐ User Defined Radius

Bend Radius: 6 in Number of Miter Cuts:

Bend Red Ratio: 1.5 Number of Bend Segments:

Dimension from "From Node" to "Tangent Intersection Point"

Delta X: 0 ft

Delta Y: 0 ft

Delta Z: -4-3-3/16 ft

Abs: 4'3-3/16" ft

Number of Intermediate Nodes: 0

Max Spacing: 0 ft

Dimension from "Tangent Intersection Point" to "Next Node"

Delta X: 4' ft

Delta Y: 0 ft

Delta Z: 0 ft

Abs: 4' ft

Flanged Ends

☐ Near End ☐ Far End

St Factor and Flex Factor

for "From Node":

for "Bend":

Bend Flex Factor:

Restraint Attachment Point on Bend Centerline

☐ Near ☒ Mid ☐ Far Angle: 0 Deg

Pipe Size

Nominal Diam: 4 in Pipe Sch: STD

Pipe size can be changed only from Pipe Properties Screen

☐ Weight Off ☐ Buoyancy calculation

OK Cancel Apply Help

- 5 Click **OK** to save the elbow data and display the Graphic View shown in Figure 2.3.3-16.

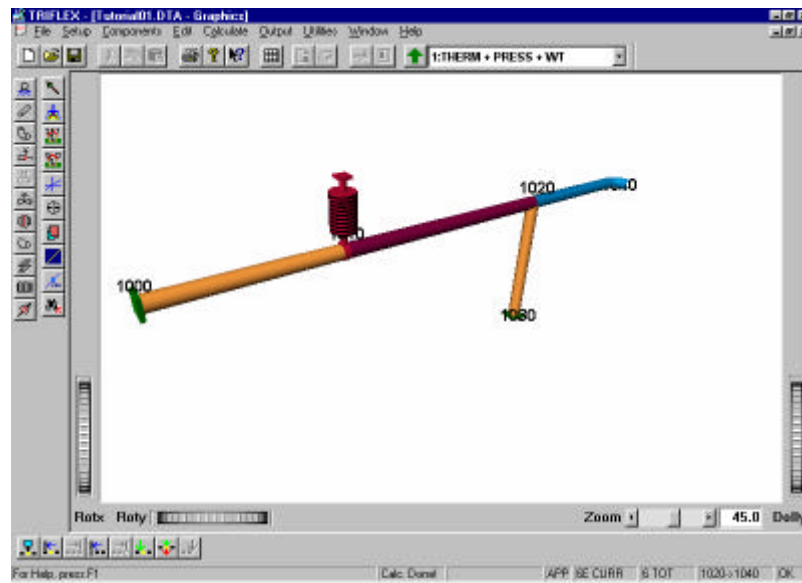


Figure 2.3.3-16 Node 1040, Elbow Data Graphic Display

2.3.6 Branch Connection

2.3.3.6 Branch Connection Data Point 1050

1. Click the **Branch Connection** Icon on the **Components** toolbar to display the **Branch Connection Data** dialog as shown in Figure 2.3.3-17. The node numbers of the node points are assigned by TRIFLEX automatically. If the user wishes a different labeling of the node points for the Branch Connection, use the pull down menu to select the node number or type in the node number you wish.

Notice that the Branch Connection Geometry defaults to “**Welding Tee S.I. Only** $T_c > 1.5*t$ ”. TRIFLEX® Windows will default to this selection unless you click on another branch connection type.

2. Tab to or use the mouse to move to the **Delta X** field of the **Dimensions from “From Node” to “To Node”** field group. Enter **4’** in the **Delta X** field to indicate that the pipe segment is 4 foot 0 inches long and that the direction from the **From Node** to the **To Node** is in the plus X direction as shown in Figure 2.3.3-17.

Data Point 1050 is now defined as the midpoint of a Welding Tee Branch

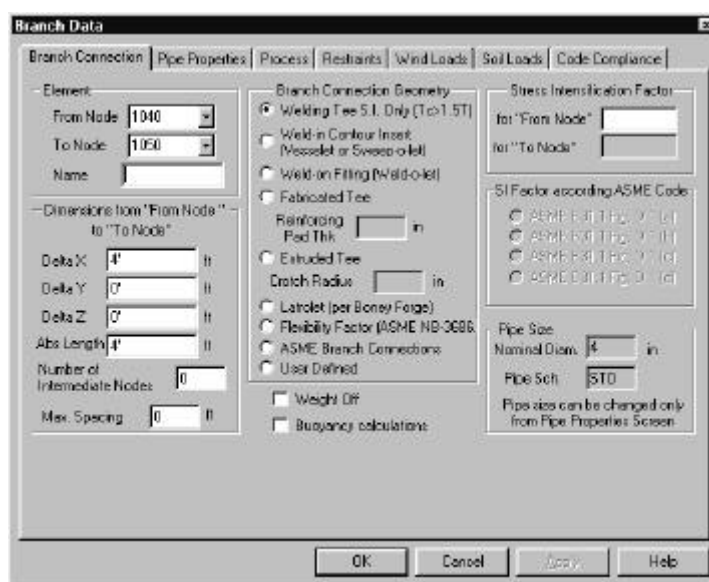


Figure 2.3.3-17 Node 1050, Branch Dialog, Branch Connection Tab

Connection and will be intensified accordingly. With the indication of the type of Branch Connection, TRIFLEX® Windows can calculate the proper stress intensification per the Piping Code specified on the **Modeling Default** dialog under the **Setup**.

4. Select **Restraint** Tab as shown in Figure 2.3.3-18.

In the **Translational Restraint Group** field group, click on the **Check Box** for the **+Y** restraint.

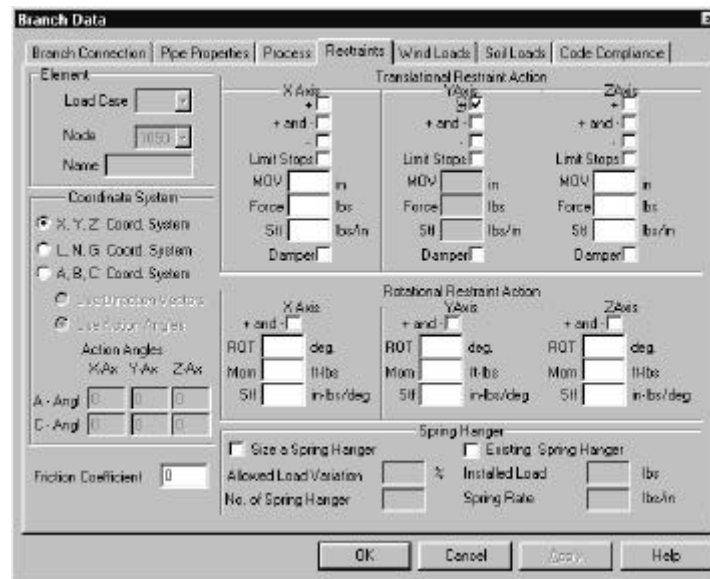


Figure 2.3.3-18 Node 1050, Branch Connection Dialog, Restraint Data Tab

Figure 2.3.3-19 Node 1050, Branch Connection Dialog Click **OK** to save the pipe and branch connection data and display the Graphic View shown in Figure 2.3.3-19.

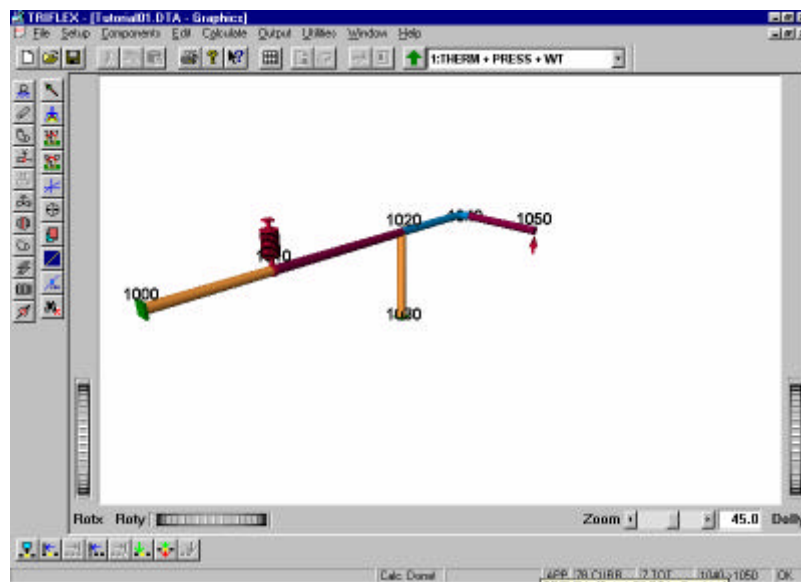


Figure 2.3.3-19 Node 1050, Branch Connection Graphic Display

2.3.3.7 Valve Data Point 1060

Please note that the piping system example calls for a flanged valve to be entered with a preceding segment of pipe between data point 1050 and data point 1060. You may enter the preceding pipe as one data point and the flanged valve as another data point, or you may enter both in one data point as we have done in the following coded dialogs. If desired, the user could also model each flange separately with one data point each and the valve without flanges with another data point. TRIFLEX Windows provides substantial modeling flexibility for the user.

To model the flanged valve with a preceding segment of pipe on one data point dialog, do the following:

1. Click on the **Valve** Icon on the **Component** toolbar. TRIFLEX will display the **Valve Data** dialog as shown in Figure 2.3.3-20.
2. Tab to or use the mouse to move to the **Delta X** field of the **Dimensions from “From Node” to “To Node”** field group. Enter **5’-10-1/8”** in the **Delta X** field to indicate that the total length of the pipe segment and the flanged valve is 5 foot 10 1/8 inches long and that the direction from the **From Node** to the **To Node** is in the plus X direction as shown in Figure 2.3.3-20.
3. In the **Valve Type** field group, click on the radio button for **Flanged Valve**.
4. In **Valve Data** field group, TRIFLEX will display **“Flanged AAAT Std Valve”** as the default. For the example problem, a flanged gate valve with weld-neck flanges has been specified. Therefore, click on the pull down menu in the Valve Type field and select the **“Flanged Gate Valve”**. The class of 150 for the flanges is correct, so no action on the user’s part is required. The other data displayed in the Valve Data field group is displayed for the user to know what data TRIFLEX is using.
5. In the **Flange Data** field group, TRIFLEX will display **“AAAT Std Flanges”** as the default. Click on the pull down menu in the Flange Type field and select the **“Weld Neck Flange”**. The other data displayed in the Flange Data field group is displayed for the user to know what data TRIFLEX is using. In addition, TRIFLEX defaults to flanges on both ends as shown in the check boxes below the flange data.
6. Below the flange data in the **“Delta Dimension Coded To”** field group, TRIFLEX allows the user to specify the location of the data point on the valve. For our example, we will accept the TRIFLEX default of locating the data point at the weld point on the far end of the valve

Figure 2.3.3-20 Node 1060, Valve Dialog, Valve Data Tab

Click **OK** to save the pipe and valve data and display the Graphic View shown in Figure 2.3.3-21.

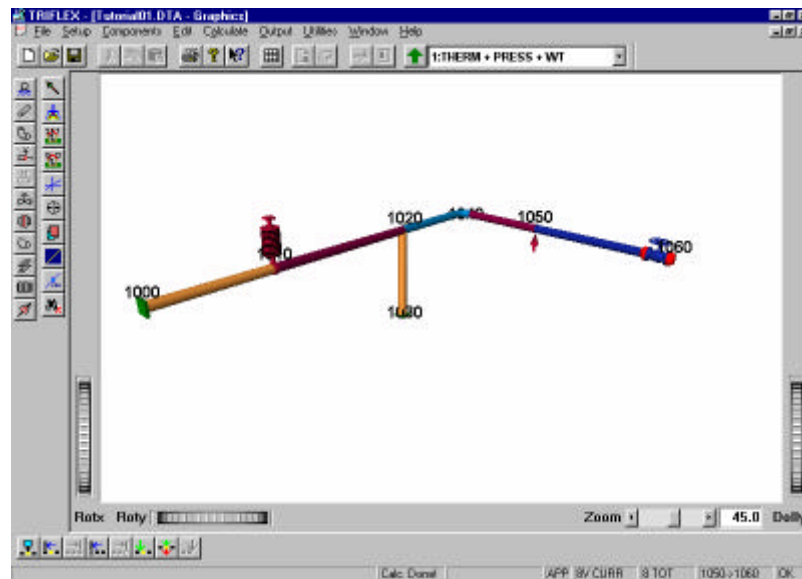


Figure 2.3.3-21 Graphic of Node 1060, Valve Dialog, Valve Data Tab

2.3.3.8 Pipe & Anchor Data Points 1060 through 1070

1. Click on the **Pipe** Icon on the **Component** toolbar.
2. Tab to or use the mouse to move to the **Delta X** field of the **Dimensions from "From Node" to "To Node"** field group. Enter **-3'-3-3/8"** in the **Delta X** field to indicate that the pipe segment is 3 foot 3 3/8 inches long and that the direction from the **From Node** to the **To Node** is in the plus X direction as shown in Figure 2.3.3-22.
3. Click **OK** to save the pipe data.

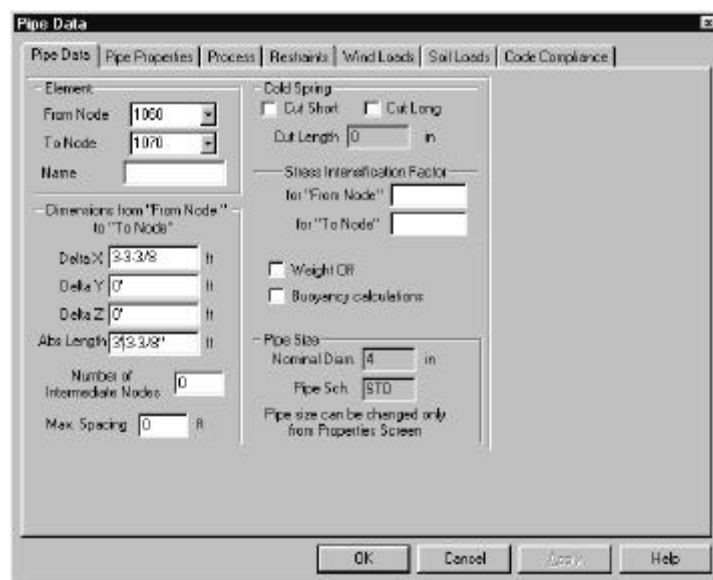


Figure 2.3.3-22 Node 1070, Pipe Dialog, Pipe Data Tab

4. Click on the **Anchor** Icon on the **Component** toolbar.
5. Verify that the number **1070** is in the **Node Label** field of the **Element** field group for the anchor as shown in Figure 2.3.3-23.
6. Click **OK** to save the anchor data and display the Graphic View shown in Figure 2.3.3-24.

The 'Anchor Data' dialog box is shown with the 'Type Location' tab selected. It contains the following fields and options:

- Wind Loads:** Type/Location (dropdown), Initial Mono/Rate (dropdown), Pipe Properties (dropdown), Process (dropdown).
- Element:** Node Label (1070), Name (text field).
- Absolute Coordinates:** X (0), Y (0), Z (0).
- Type:** Radio buttons for 'Anchor is Totally Rigid' (selected), 'Anchor is Totally Free', '*** Axis Rigid', and 'User Defined Stiffness'.
- Translational Stiffness:** X (Fixed), Y (Fixed), Z (Fixed).
- Rotational Stiffness:** X (Fixed), Y (Fixed), Z (Fixed).
- Free along *** axis when Weight Analysis Processed for Spring Hanger Design:** (checkbox, unchecked).

Buttons at the bottom: OK, Cancel, Apply, Help.

Figure 2.3.3-23 Node 1070, Anchor Dialog, Type Location Tab

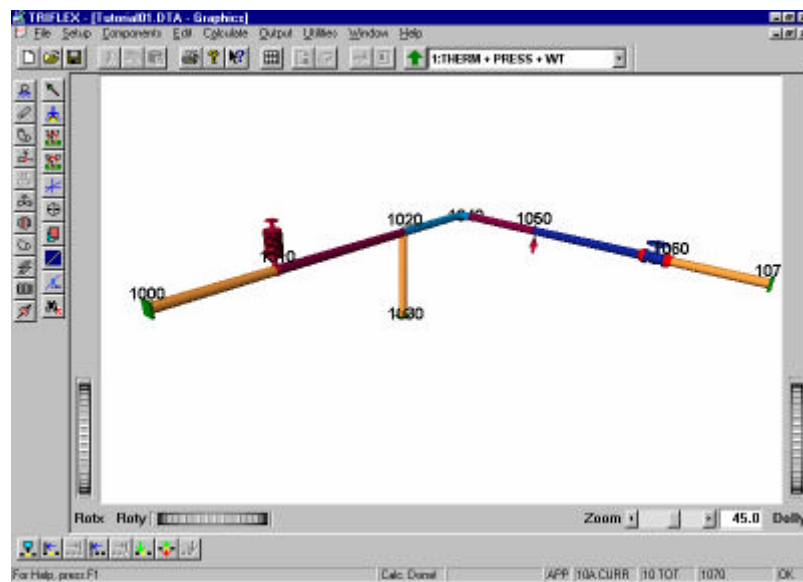


Figure 2.3.3-24 Node 1070, Pipe & Anchor Data Graphic Display

1. Click on the **Pipe** Icon on the **Component** toolbar.
2. Use the pull down menus to select the correct node numbers for the **From Node - 1050** and the **To Node - 1080**.
3. Tab to or use the mouse to move to the **Delta Z** field of the **Dimensions from "From Node" to "To Node"** field group. Enter **-5'-2-1/8"** in the **Delta Z** field to indicate that the pipe segment is 5 foot 2 1/8 inches long and that the direction from the **From Node** to the **To Node** is in the negative Z direction as shown in Figure 2.3.3-25.

Figure 2.3.3-24 Node 1070, Pipe & Anchor Data Graphic Display

Pipe Data

Pipe Data | Pipe Properties | Process | Restraints | Wind Loads | Soil Loads | Code Compliance

Element:
 From Node: 1050
 To Node: 1080
 Name:

Dimensions from "From Node" to "To Node":
 Delta X: 0" ft
 Delta Y: 0" ft
 Delta Z: -5'-2-1/8" ft
 Abs Length: 5'-2-1/8" ft
 Number of Intermediate Nodes: 0
 Max Spacing: 0 ft

Cold Spring:
☐ Cut Short ☐ Cut Long
 Cut Length: 0 in

Stress Intensification Factor:
 for "From Node":
 for "To Node":

☐ Weight OR
☐ Buoyancy calculations

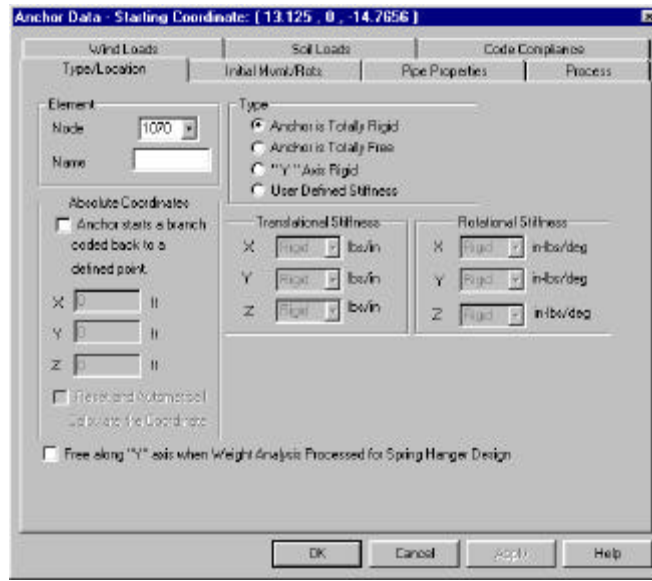
Pipe Size:
 Nominal Diam: 4 in
 Pipe Sch: STD
 Pipe size can be changed only from Properties Screen

OK Cancel Apply Help

4. Click **OK** to save the pipe data.
5. Click on the **Anchor** Icon on the **Component** toolbar.
6. Verify that the number **1080** is in the **Node Label** field of the **Element** field group for the anchor as shown in Figure 2.3.3-26.
7. Click **OK** to save the anchor data and display the Graphic View shown in Figure 2.3.3-27.

Figure 2.3.3-25 Node 1080, Pipe Data Sheet, Pipe Data Tab

The system should graphically look like Figure 2.3.3-27. To display the coordinates axis system on the model, click on the **Toggle Axis** Icon on the **Graphic Toolbar**.



The dialog box is titled "Anchor Data - Starting Coordinate: (13.125 , 0 , -14.7656)". It has four tabs: "Wind Loads", "Soil Loads", "Code Compliance", and "Type/Location". The "Type/Location" tab is selected. It contains the following fields and options:

- Element:**
 - Node: 1080
 - Name: (empty)
- Type:**
 - ☒ Anchor is Totally Rigid
 - ☐ Anchor is Totally Free
 - ☐ "Y" Axis Rigid
 - ☐ User Defined Stiffness
- Absolute Coordinates:**
 - ☐ Anchor starts a branch coded back to a defined point.
 - X: 0 ft
 - Y: 0 ft
 - Z: 0 ft
 - ☐ Invert and Automatch
 - ☐ Calculate the Coordinates
 - ☐ Free along "Y" axis when Weight Analysis Processed for Spring Hanger Design
- Translational Stiffness:**
 - X: Rigid lbs/in
 - Y: Rigid lbs/in
 - Z: Rigid lbs/in
- Rotational Stiffness:**
 - X: Rigid in-lbs/deg
 - Y: Rigid in-lbs/deg
 - Z: Rigid in-lbs/deg

Buttons at the bottom: OK, Cancel, Apply, Help.

Figure 2.3.3-26 Node 1080, Anchor Data Sheet, Type Location Tab

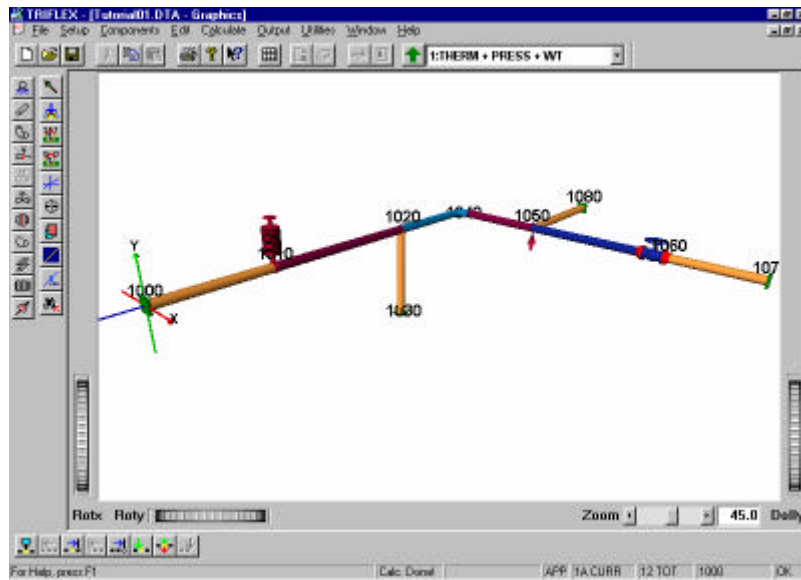


Figure 2.3.3-27 Node 1080, Pipe & Anchor Data Graphic Display

2.3.4 Executing a Static Analysis

To process a TRIFLEX® Windows analysis of the piping system you just entered, click on the **Green Arrow** icon in the **Main Menu** or from the **Setup** menu, select the **Basic** option as shown in Figure 2.3.4-1.

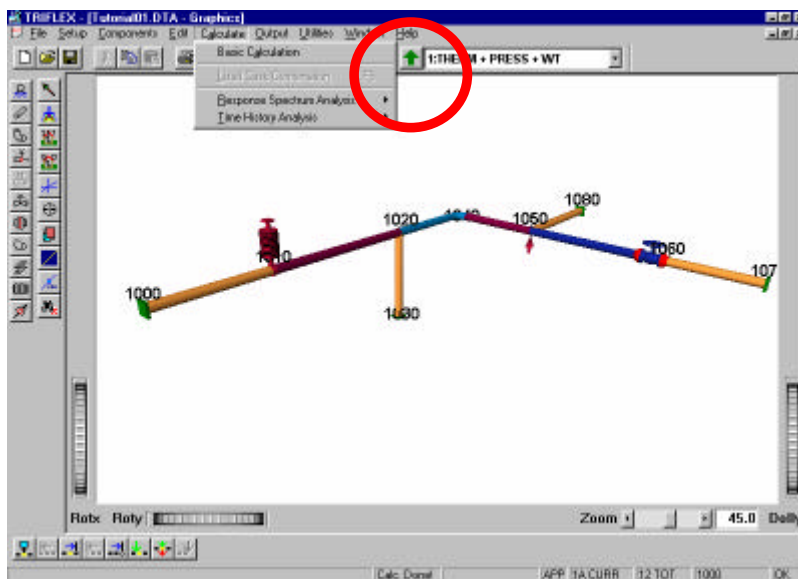


Figure 2.3.4-1 Main Screen, Calculate Pull-Down Menu

Note: A case number must be filled in before TRIFLEX® Windows will perform the stress calculations.

Once TRIFLEX has been instructed to process the analysis, the program will begin executing the stress calculations. The status of the calculations will be displayed in the TRIFLEX® Windows screen.

While the calculation is in progress, the **Calculation Ready/Stop** Icon will be displayed as a red stop sign as shown in Figure 2.3.4-2. To stop the calculation process, click the **Calculation Ready/Stop** Icon and the calculations will be immediately aborted.

Upon completion of the calculation process, the **Calculation Ready/Stop** Icon will be returned to the green arrow ready state as shown in Figure 2.3.4-1.

2.3.5 View Run Output

To view the results of the stress calculations in spreadsheet format, do the following:

1. From the **Output Pull Down** menu, select **View Results**. See Figure 2.3.5-1 below for this menu. The TRIFLEX® Windows calculation results will be displayed as shown in Figure 2.3.5-2.

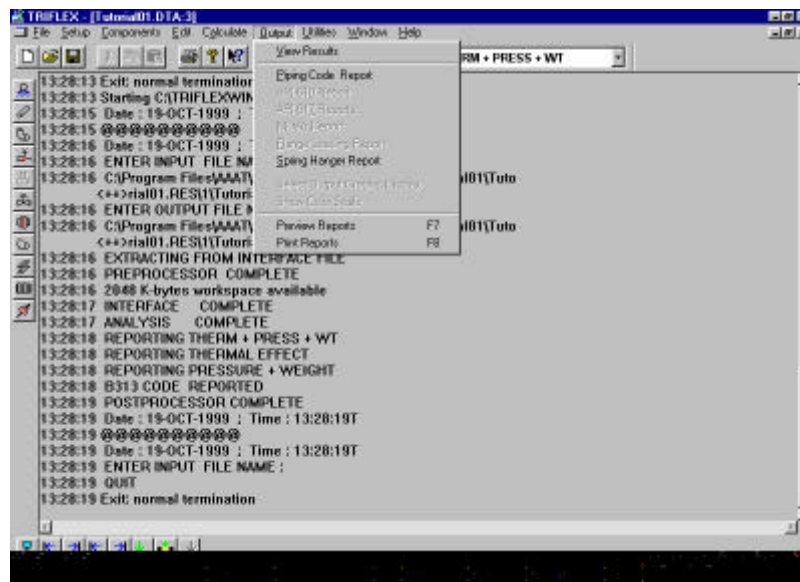
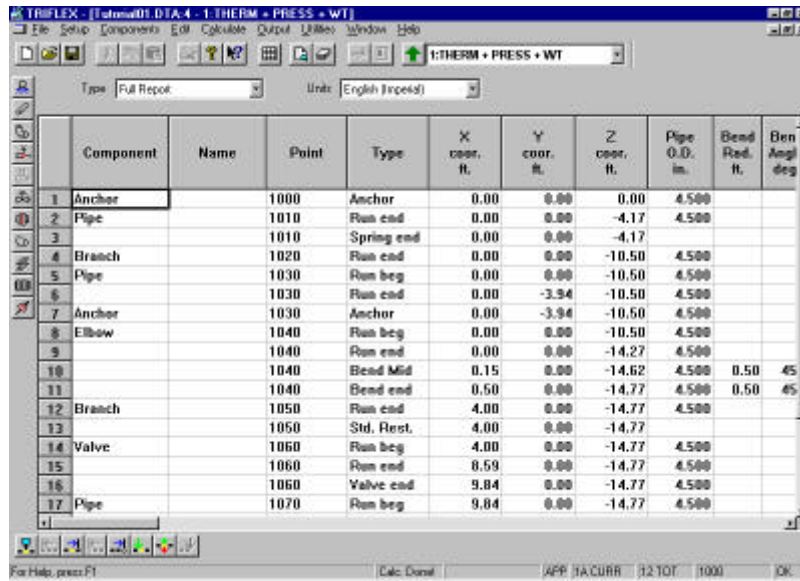


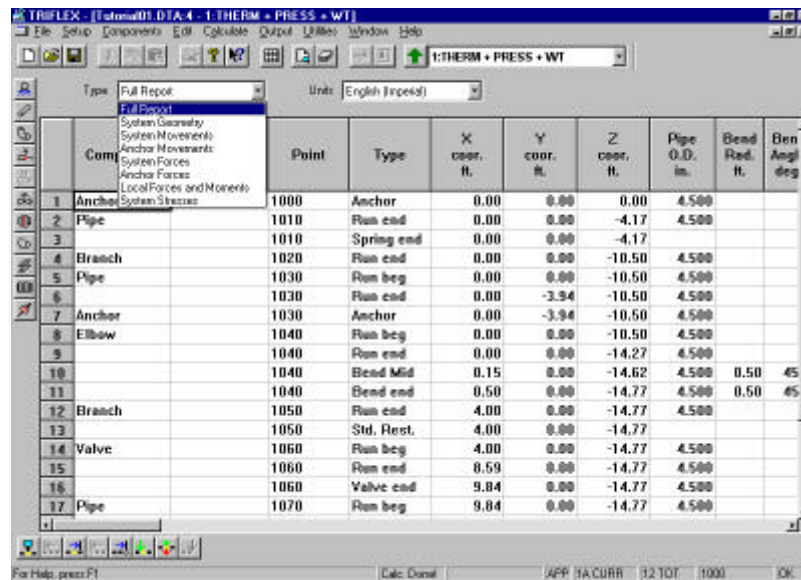
Figure 2.3.5-1 Output Pull-Down Menu

2. Select the **Load Case** that you wish to view using the **Load Case pull down** menu as shown in Figure 2.3.5-2.
3. Select the report that you wish to view using the **Type Report Selector pull down** menu as shown in Figure 2.3.5-3.



| | Component | Name | Point | Type | X coord. ft. | Y coord. ft. | Z coord. ft. | Pipe O.D. in. | Bend Rad. ft. | Ben Angl deg |
|----|-----------|------|-------|------------|--------------------|--------------------|--------------------|---------------------|---------------------|--------------------|
| 1 | Anchor | | 1000 | Anchor | 0.00 | 0.00 | 0.00 | 4.500 | | |
| 2 | Pipe | | 1010 | Run end | 0.00 | 0.00 | -4.17 | 4.500 | | |
| 3 | | | 1010 | Spring end | 0.00 | 0.00 | -4.17 | | | |
| 4 | Branch | | 1020 | Run end | 0.00 | 0.00 | -10.50 | 4.500 | | |
| 5 | Pipe | | 1030 | Run beg | 0.00 | 0.00 | -10.50 | 4.500 | | |
| 6 | | | 1030 | Run end | 0.00 | -3.94 | -10.50 | 4.500 | | |
| 7 | Anchor | | 1030 | Anchor | 0.00 | -3.94 | -10.50 | 4.500 | | |
| 8 | Elbow | | 1040 | Run beg | 0.00 | 0.00 | -10.50 | 4.500 | | |
| 9 | | | 1040 | Run end | 0.00 | 0.00 | -14.27 | 4.500 | | |
| 10 | | | 1040 | Bend Mid | 0.15 | 0.00 | -14.62 | 4.500 | 0.50 | 45 |
| 11 | | | 1040 | Bend end | 0.50 | 0.00 | -14.77 | 4.500 | 0.50 | 45 |
| 12 | Branch | | 1050 | Run end | 4.00 | 0.00 | -14.77 | 4.500 | | |
| 13 | | | 1050 | Std. Rest. | 4.00 | 0.00 | -14.77 | | | |
| 14 | Valve | | 1060 | Run beg | 4.00 | 0.00 | -14.77 | 4.500 | | |
| 15 | | | 1060 | Run end | 8.59 | 0.00 | -14.77 | 4.500 | | |
| 16 | | | 1060 | Valve end | 9.84 | 0.00 | -14.77 | 4.500 | | |
| 17 | Pipe | | 1070 | Run beg | 9.84 | 0.00 | -14.77 | 4.500 | | |

Figure 2.3.5-2 Output Load Case Pull-Down Menu



| | Component | Name | Point | Type | X coord. ft. | Y coord. ft. | Z coord. ft. | Pipe O.D. in. | Bend Rad. ft. | Ben Angl deg |
|----|-----------|------|-------|------------|--------------------|--------------------|--------------------|---------------------|---------------------|--------------------|
| 1 | Anchor | | 1000 | Anchor | 0.00 | 0.00 | 0.00 | 4.500 | | |
| 2 | Pipe | | 1010 | Run end | 0.00 | 0.00 | -4.17 | 4.500 | | |
| 3 | | | 1010 | Spring end | 0.00 | 0.00 | -4.17 | | | |
| 4 | Branch | | 1020 | Run end | 0.00 | 0.00 | -10.50 | 4.500 | | |
| 5 | Pipe | | 1030 | Run beg | 0.00 | 0.00 | -10.50 | 4.500 | | |
| 6 | | | 1030 | Run end | 0.00 | -3.94 | -10.50 | 4.500 | | |
| 7 | Anchor | | 1030 | Anchor | 0.00 | -3.94 | -10.50 | 4.500 | | |
| 8 | Elbow | | 1040 | Run beg | 0.00 | 0.00 | -10.50 | 4.500 | | |
| 9 | | | 1040 | Run end | 0.00 | 0.00 | -14.27 | 4.500 | | |
| 10 | | | 1040 | Bend Mid | 0.15 | 0.00 | -14.62 | 4.500 | 0.50 | 45 |
| 11 | | | 1040 | Bend end | 0.50 | 0.00 | -14.77 | 4.500 | 0.50 | 45 |
| 12 | Branch | | 1050 | Run end | 4.00 | 0.00 | -14.77 | 4.500 | | |
| 13 | | | 1050 | Std. Rest. | 4.00 | 0.00 | -14.77 | | | |
| 14 | Valve | | 1060 | Run beg | 4.00 | 0.00 | -14.77 | 4.500 | | |
| 15 | | | 1060 | Run end | 8.59 | 0.00 | -14.77 | 4.500 | | |
| 16 | | | 1060 | Valve end | 9.84 | 0.00 | -14.77 | 4.500 | | |
| 17 | Pipe | | 1070 | Run beg | 9.84 | 0.00 | -14.77 | 4.500 | | |

Figure 2.3.5-3 Output Type Pull-Down Menu

To view Code Compliance Report

- From the **Output Pull Down** menu, select **Piping Code Report** similar to that shown in Figure 2.3.5-1. The TRIFLEX® Windows calculation results will be displayed as shown in Figure 2.3.5-4.

| | FROM | TO | ALLOWABLE HOT STRESS WITH WELD F. | ALLOWABLE COLD STRESS psi | ALLOWABLE HOT STRESS psi | STRESS RANGE REDUCTION FACTOR | OCCASIONAL FATIGUE FACTOR | Y COEFFICIENT | MI TOLER |
|---|------|------|--|------------------------------------|-----------------------------------|--|---------------------------------|------------------|-------------|
| 1 | 1000 | 1000 | 20000 | 20000 | 20000 | 1.00 | 1.20 | 0.40 | 12.5001 |

| | DATA POINT | NODE LOCATION | WALL THICKNESS DESIGN in. | WALL THICKNESS REQUIRED in. | SUSTAINED STRESS ACTUAL psi | SUSTAINED STRESS ALLOWED psi | EXPANSION STRESS ACTUAL psi | EXPANSION STRESS ALLOWED psi |
|----|---------------|------------------|------------------------------------|--------------------------------------|--------------------------------------|---------------------------------------|--------------------------------------|---------------------------------------|
| 1 | 1000 | Anchor | 0.237 | 0.009 | 2058 | 20000 | 1898 | 47942 |
| 2 | 1010 | Run end | 0.237 | 0.009 | 1835 | 20000 | 2636 | 48165 |
| 3 | 1010 | Spring end | | | | | | |
| 4 | 1020 | Run end | 0.237 | 0.009 | 2034 | 20000 | 15530 | 47966 |
| 5 | 1030 | Run beg | 0.237 | 0.009 | 2388 | 20000 | 27680 | 47612 |
| 6 | 1030 | Run end | 0.237 | 0.009 | 522 | 20000 | 24496 | 49478 |
| 7 | 1030 | Anchor | | | | | | |
| 8 | 1040 | Run beg | 0.237 | 0.009 | 654 | 20000 | 21511 | 49346 |
| 9 | 1040 | Run end | 0.237 | 0.009 | 419 | 20000 | 21628 | 49581 |
| 10 | 1040 | Bend Mid | 0.237 | 0.009 | 439 | 20000 | 22098 | 49561 |
| 11 | 1040 | Bend end | 0.237 | 0.009 | 364 | 20000 | 18067 | 49636 |

Figure 2.3.5-4 Output Code Compliance Report

To view the piping model output graphically,

- Click on the **OutPut Display** icon in the **Main Menu** as shown in Figure 2.3.5-5 or, from the **Output Pull Down** menu, select **Select Output Graphic Display** similar to that shown in Figure 2.3.5-1. An OutPut display screen will appear in the middle of the screen.
- In the **Output Display** screen, click on the **Display Pull Down** menu as shown in Figure 2.3.5-6 to select the calculated output data that you wish to view.
- If you select deflections, rotations, forces or moments, you must then select the **Line of Action** that you wish. Under Line of Action, TRIFLEX will default to Resultant values unless you specify another category. Then click OK.

If you select any of the stresses calculated by TRIFLEX, then you must select either **Absolute Value** or **Sign (+/-)** from the Stress Display group. Under the Stress Display group, TRIFLEX will default to Absolute values unless you specify Sign (+/-). Then click OK.

Note: If your piping model does not appear on the screen at this point, then press Control + Tab to toggle between all screens available describing the piping system. Stop when you see the piping model. Alternatively, you can click on the Spreadsheet Icon to toggle between the spreadsheet view and the piping model.

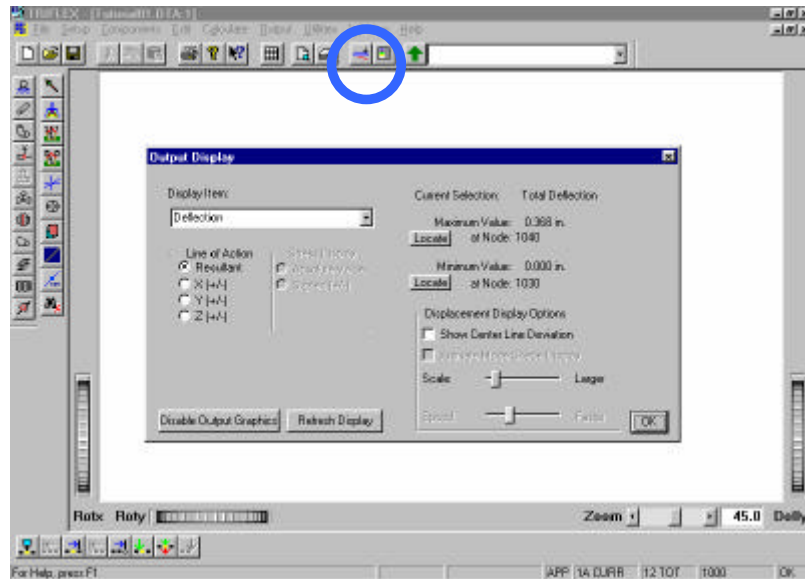


Figure 2.3.5-5 Output Graphic, Output Display Menu

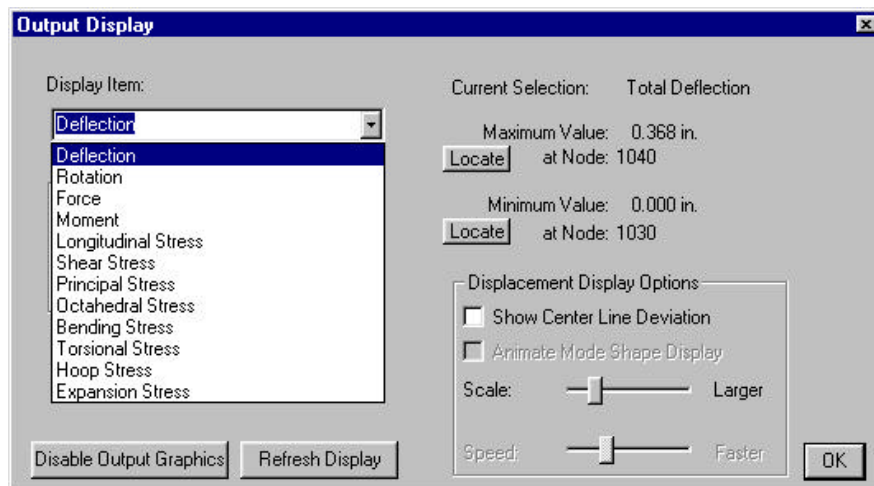


Figure 2.3.5-6 Output Display Screen, Display Pull Down Menu

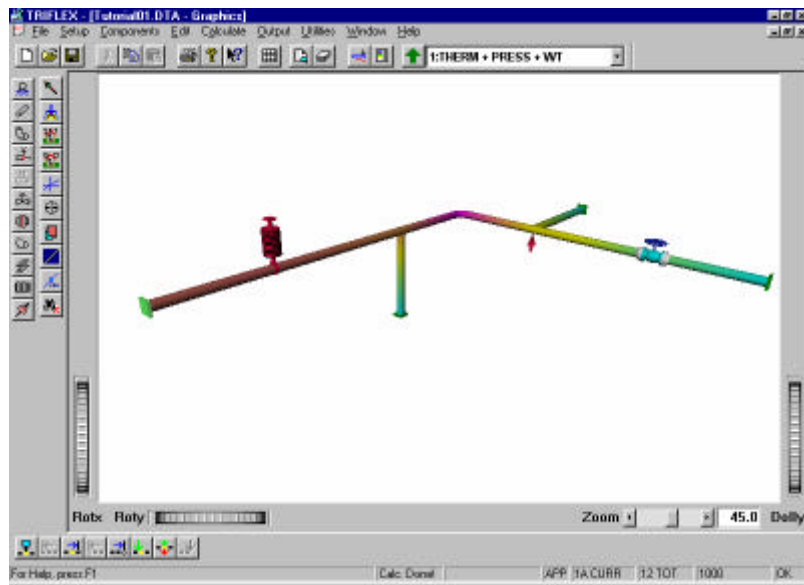


Figure 2.3.5-7 Output Graphic Screen

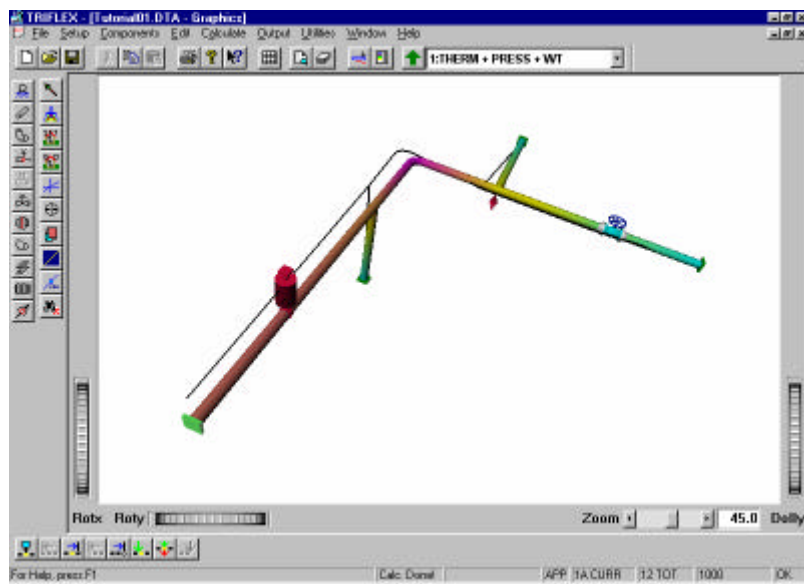


Figure 2.3.5-8 Output Graphic – Deformed Shape

To view the piping model with a superimposed deformed shape,

8. In the **Output Display** screen shown in Figure 2.3.5-6, click on the **Display Pull Down** menu and select **Deflection**.
9. Then on the **Output Display** screen, click on the check box for **Show Center Line Deviation** and enter a number in the **Scale** field indicating the multiplier factor to be applied to the deflection shown on the model. Then click OK. A screen showing the deformed piping model will then appear as shown in Figure 2.3.5-8.

2.3.6 Printing

TRIFLEX® Windows has an extremely easy to use facility for printing output reports and screens.

1. From the **Output Pull Down** menu, select **Preview Report** or **Print Report** similar to that shown in Figure 2.3.6-1. The Report Print (Print Static / Dynamic Reports) screen will then appear.

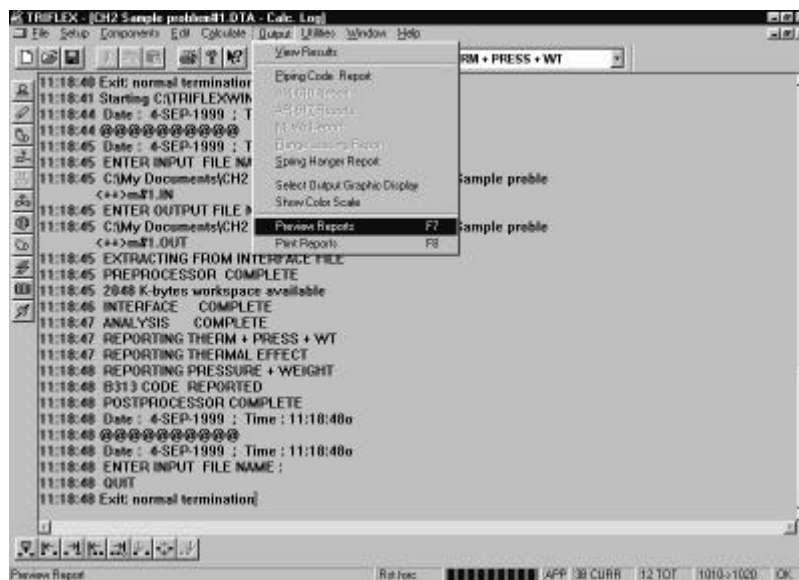


Figure 2.3.6-1 Output Pull-Down Menu

2. In **Print Report** or **Preview Report** screen, select the **Loading Case** and the reports from the **Available Report** group by placing a check in the box adjacent to each desired report as shown in Figure 2.3.6-2.
3. A **Preview Report** sample screen is shown in Figure 2.3.6-3.

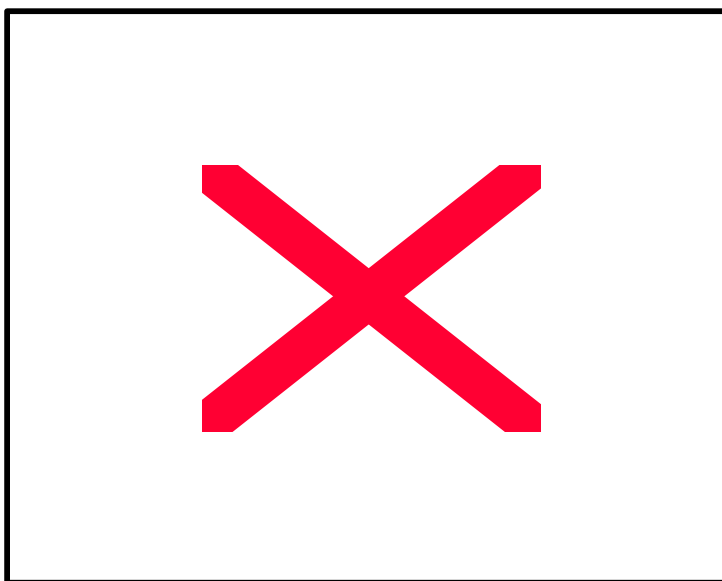


Figure 2.3.6-2 Report Print Menu

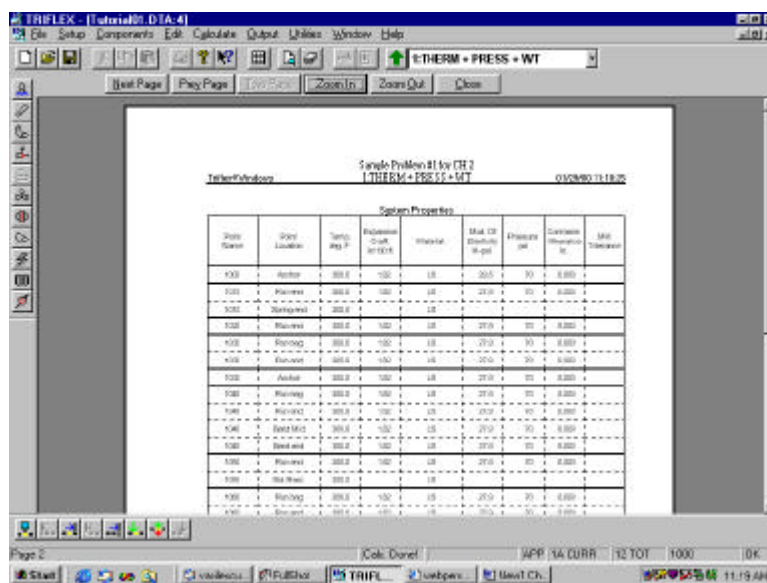


Figure 2.3.6-3 Report Print Preview Sample

This completes our TRIFLEX® Windows tutorial. We have by no means covered all the capabilities of the program, but you should have a better understanding of the general approach for building a piping model, executing an analysis and reviewing the results.

To exit TRIFLEX® Windows, click **EXIT** under **FILE**. Please remember before you exit TRIFLEX to save your model. To do so, from the **File Pull Down** menu, select **Save As** similar to the procedure used in most Windows programs.

APPENDIX A**TRIFLEX® Windows Command and Shortcut Keys**
(Graphics Mode)**COMMANDS**

- Help
- Worksheet Toggle
- Start Calculation
- Print Report Preview
- Print Report

- Move to End
- Edit Current Component
- Move to First Component
- Insert
- Move to Next Component
- Move to Previous Component

- Copy
- Cut
- Delete
- New
- Open
- Paste
- Print
- Save
- Undo

SHORTCUTS

F1 (not active currently)

F4

F5

F7

F8

END

F9

HOME

INS

PGDN

PGUP

CTRL + C

CTRL + X

DEL

CTRL + N

CTRL + O

CTRL + V

CTRL + P

CTRL + S

CTRL + Z

- | | |
|---|-------------|
| • “Arrow” moves model | ARROW KEYS |
| • Bring up “Start” | CTRL + ESC |
| • Capture display to Clipboard | ALT + PRTSC |
| • Change pointer/manipulator | ESC |
| • Change pointer to manipulator | ALT + SHIFT |
| • Change to manipulator (temporary) | ALT |
| • Display all available windows | ALT + ESC |
| • Manipulator moves model | SHIFT |
| • Next (toggle between graphics & spreadsheet input) | CTRL + F6 |
| • Toggle through all available windows | ALT + TAB |

APPENDIX B

Procedure to read

* .IN file generated in TRIFLEX Windows into TRIFLEX DOS

The .IN file is created by TRIFLEX Windows during of the run session of the model. This file will be saved with the same name as the name of piping model (by default the name is “untitled”) and has .IN extension.

TRIFLEX Windows will create a .IN file for each defined load case.

Locations where the .IN are saved are: **drive:\...\name of the model\1** for the load case 1, **drive:\...\name of the model\2** for the load case 2 and so on.

In order to read this files in TRIFLEX DOS follow the next steps:

1. Copy the .IN file to the folder Drive:\AAALIB (the default directory for TRIFLEX DOS installing) to the directory where the TRIFLEX DOS was installed.

```
TRIFLEX Release 2.7.1 - Primary Options Menu
Copyright 1986-99 AAA Technology & Specialties Co. Inc.
Houston, Texas 77041-5114, USA
Current Job Tree: c:\triflex
Current Job Code: TESTTR
```

- | | |
|---|-----------------------------|
| 1. Job Definition | Select Job / Setup New Job |
| 2. Data Entry | Pipe Config. Stats/Dynamics |
| 3. Perform TRIFLEX Analysis | |
| 4. Display / Plot / Print TRIFLEX Results | |
| 5. TRIFLEX Utilities Menu | |

Select an option (1 -> 5) or [Esc] to return to DOS

2. Run TRIFLEX DOS and select option 5. TRIFLEX Utilities Menu

2. Select Option 3 – Convert into Triflex Format

Copyright 1986-98 AAA Technology & Specialties Co. Inc., Houston, Texas 77041

Current Job Code: TESTTR

1. Update the TRIFLEX Physical Properties Database
 2. Update the TRIFLEX Structural Properties Database
 - 3.** Convert into TRIFLEX Format
 4. Change Display Colors
 5. Display Hardware Characteristics
 6. Execute a DOS Command
 7. Browse Current Output File
 8. Customize graphics (Adapter/Printer)

Select an option (1 - 8) or [Esc] to return to the Primary Options Menu

3. Select Option 5 – Convert Triflex Keyword Files To Triflex Node Data

For Job Code: TESTTR

Job Case Number: 1

1. Convert TRIFLEX Results Brought from a Remote System to the PC.
 2. Convert CADPIPE(tm) Connectivity File to TRIFLEX Node Data.
 3. Convert Intergraph PDS(tm) Neutral File to TRIFLEX Node Data.
 4. Convert CALMA(tm) Import File to TRIFLEX Node Data.
 - 5.** Convert TRIFLEX Keyword File to TRIFLEX Node Data.
 6. QUIT

Select an option (1 - 6) or [Esc] to return to the Primary Options menu.

3. Using arrows button select which file you want to convert (in our case TESTTR) and press F10
5. Press any key

TRIFLEX Conversion: From TRIFLEX Keywords
Keywords Work Path: C:\AAALIB

Job Code: TESTTR

Keywords File Model Name: **TESTTR**

TRIFLEX Target Job Code: TESTTR

| TRIFLEX | |
|---------------|----------|
| FLEX029 | NRC |
| TESTTR | SAMPLE01 |
| TUTORI~1 | SB1721 |
| | SOIL-2 |
| | SWMET22 |
| | TEEEEST1 |
| | TESTDIN1 |
| | TESTK |
| | TESTTR |
| | THROB |
| | TRIFLE~1 |
| | TSET1.RE |

F1: Help F3: Quit Crsr Up: ↑ PgUp: ↑ 10 rows Home: Top
F4: Create TRIFLEX Job Crsr Down: ↓ PgDn: ↓ 10 rows End: Bottom
F10: Convert to TRIFLEX F5/Enter : Select From Show Window

Conversion of a AAA-Base File : C:\AAALIB\TESTTR.AAA
Establishing AAA Front End Job : c:\triflex\TESTTR\TESTTR
End of Conversion

CAUTION: Converted data may be INCOMPLETE
Scrutiny of the newly generated TRIFLEX job is advisable
Press any key to reach the Node Data Spreadsheet

6. You can see, edit run, and save the imported pipe model

| FROM | TO | NODE TYPE | PPP CHNGE | RST | SIF | 5 | 7 Nodes Last | 5 30 | Top End | 5 30 | Low High |
|------|----|--------------|--------------|-----|-----|---|------------------------------|---------------|------------|---------|-------------|
| | 5 | A | PP | | | | | | | | |
| 5 | 10 | R | | | | | F2: Search | | | | |
| 10 | 15 | B | | | | | F1: Help | F5: Block | | | |
| 15 | 20 | R | | | | | F4: Plot | F3: Quit | | | |
| 20 | 25 | B | | | | | Del: REMOVE the highlighted | | | | |
| 25 | 30 | R | | | | | F6: node from the database | | | | |
| | 30 | A | | | | | Ins: INSERT one or more | | | | |
| | | | | | | | F9: nodes at the highlight | | | | |
| | | | | | | | Enter: DISPLAY detail screen | | | | |
| | | | | | | | F10: for highlighted node | | | | |
| | | | | | | | ↑: Up 1 row | ↓: Down 1 row | | | |
| | | | | | | | Home: Move to top row | | | | |
| | | | | | | | End: Move to bottom row | | | | |
| | | | | | | | Pg Up: Move up 1 page | | | | |
| | | | | | | | Pg Dn: Move down 1 page | | | | |

APPENDIX B

Procedure to read

* .IN file generated in TRIFLEXDOS into TRIFLEX Windows

It is possible to import .IN files and .JOB files written by TRIFLEXDOS in a new empty file opened by TRIFLEX Window.

In order to read these files in TRIFLEX DOS follow the next steps:

1. Start TRIFLEX Windows program or if it is already started and a file opened select, **File** and **New** option from Main Menu.
2. From Main Menu select **Utilities**, **Import Files** and **Triflex Keyword** option.
3. Select the directory where the .IN file is located, select the **.IN** file and press the **OPEN** button.
4. The graphic of the pipe system will become visible on the screen.

Follow the rules for editing, running and saving which are specific for TRIFLEX Windows.

APPENDIX A

Installation Instructions

For

TRIFLEX® Windows

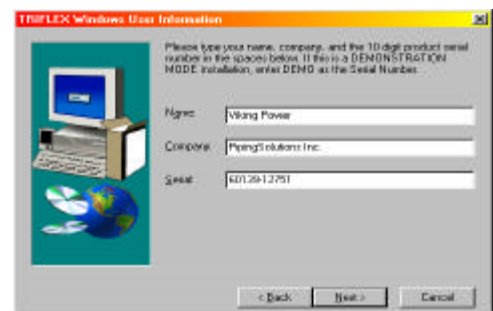
When installing on a NT machine, the installer needs to login as the NT administrator.

Insert the CD into the CD Reader, the following screen will appear.

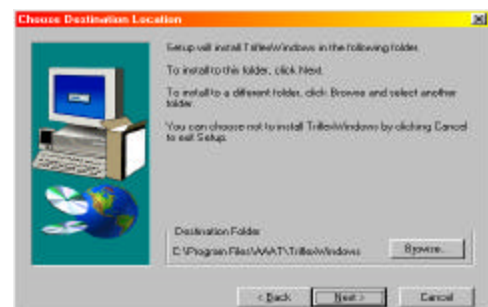
Select Install Triflex Windows and follow the instructions.



When the following screen appears enter the fields with the proper information. A proper serial number or the word DEMO must be entered before the “Next” button is made available.



Here you can specify the folder where you want TRIFLEX Windows to be installed. The default settings are:

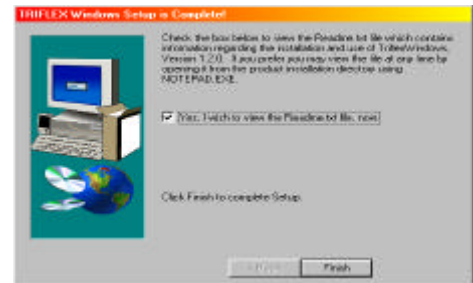


C:\ProgramFiles\PipingSolutions\TriflexWindows

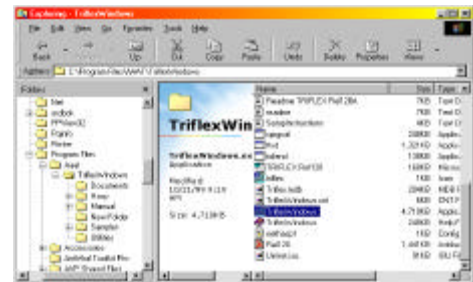
The next screen will just tell you if you have enough space to install. Hit next, to proceed with the installation. The status bar will tell you the progress of the installation



Triflex Windows is now installed, you can view the release notes by simply hitting “Finish”.



To Run TRIFLEX Windows start Windows Explorer and locate the TRIFLEX executable.



To make a shortcut, simply drag and drop the TRIFLEX icon over to your desktop.

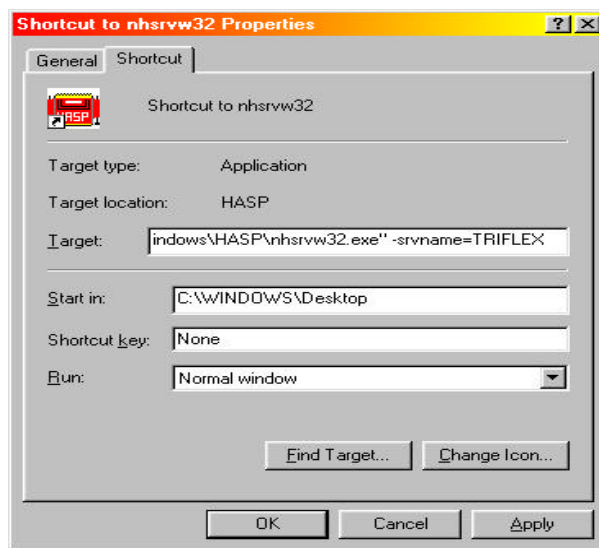
NETWORK INSTALLATION

When installing on a NT machine, the installer needs to login as the NT administrator. Network installation requires a couple of extra steps. The following works fine with Netbios and TCP/IP types of networks.

If you are unfamiliar with the inner workings of Windows and Networking, please have the IT-Department to do the following:

SERVER

It is not necessary to have a full installation of TRIFLEX Windows on the server. The HASP folder can be copied over to the server from the client. However, both SERVER and CLIENT can reside on same computer.



1. Now we need to set up the shortcut for the NHSRVW32 program. Since our Nethasp.ini (this is the user part not the server) is looking for the License Manager called "TRIFLEX". WE need to make sure we start the LM and give it a name.
2. To give the LM a name, make a shortcut to the NHSRVW32.
3. For NT-Systems, we recommend to place the NHSRVW32 Shortcut at

<SysDRV>:\WINNT\Profiles\All Users\Start Menu\Programs\Startup location. This will assure the access to TRIFLEX Application no matter

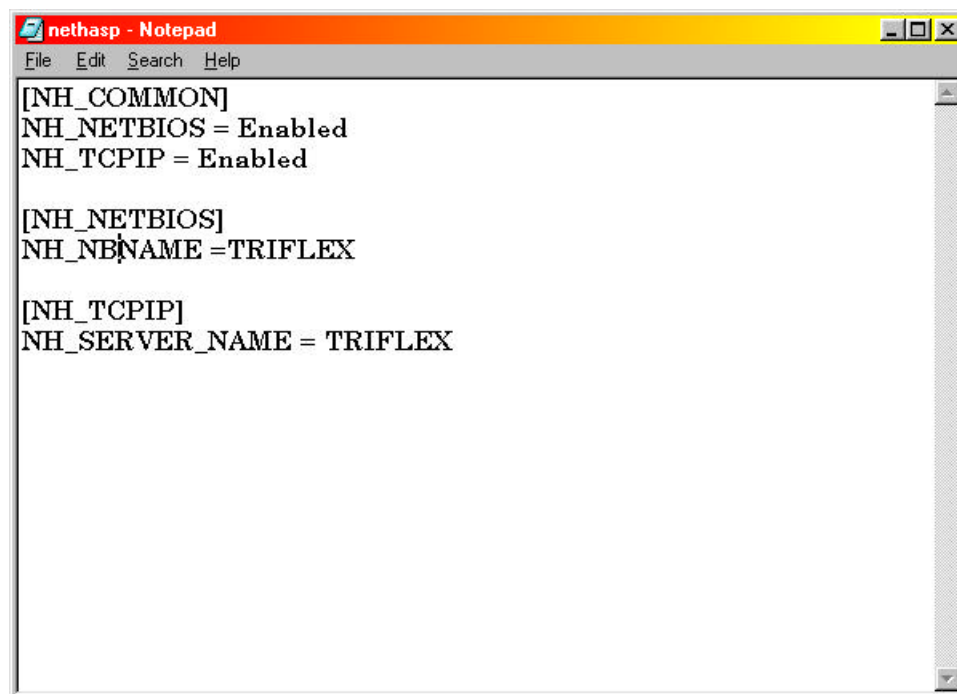
which machine was designated as dedicated server or who is working on it.

Right Click on the Icon, left click on Properties Add on to the target line –
srvname=TRIFLEX

4. Right Click on the Icon, left click on Properties Add on to the target line –
srvname=TRIFLEX

5. Add on to the target line –srvname=TRIFLEX

CLIENT



Move the **Nethasp.ini** from the HASP folder, which is located in the TRIFLEX Windows folder.

This is a typical content of the Nethasp.ini file.